

MEGSO

INTRODUCTION TO ABAQUS CAE

PRADEEP GUDLUR

MECHANICAL ENGINEERING

TEXAS A & M UNIVERSITY

4th Feb 2012, 10am to 1pm, Zachry 105C





Objectives & Outline

What you will learn from this workshop?

- How to launch ABAQUS from EOS (supercomputing facility)
- How to use the ABAQUS manual
- How to download an example file from ABAQUS manual and how to import it into ABAQUS?
- Preprocessing, running the analysis and post processing using the following examples
 - a) Example 1: 3D stress analysis
 - b) Example 2: Transient Heat transfer Analysis
 - c) Example 3: Extrusion of metal (Dynamic Explicit Analysis)
 - d) Example 4: Frequency Analysis of Gear
 - e) Example 5: Modeling of 2D particle reinforced composite
 - f) Example 6: Modeling of 3D particle reinforced composite

Introduction: some basics

File type	Purpose
.cae	<ul style="list-style-type: none"> a) All the preprocessing can be done with this b) Can be opened with ABAQUS CAE
.inp	<ul style="list-style-type: none"> a) Analysis input file. b) One of the ways to generate an input file is by using ABAQUS CAE c) Contains all the information related to the problem (ex: coordinates of nodes, element connectivity matrix, element type, material props, step, type of analysis performed, boundary conditions, loads etc). d) input file can be opened with wordpad and can be edited. It can also be imported into ABAQUS CAE. e) Using the following command input file is run: <code>abaqus job=jobname.inp</code>
.odb	<ul style="list-style-type: none"> a) Results file written by the analysis. b) Can be opened with ABAQUS viewer (opens in the visualization module) and any postprocessing can be performed.
.sta	<ul style="list-style-type: none"> a) Status file. The analysis writes incremental summaries to this file b) We can know step time, total time, CPU time, increment number, # of equilibrium iterations, severe discontinuity iterations etc in a tabular format c) Can be opened with wordpad d) You should see “The analysis has completed successfully” in the status file. otherwise analysis didn’t complete and the errors can be seen in .msg and .dat files if they exist
.msg	<ul style="list-style-type: none"> a) Message file written by the analysis b) contains any error or warning messages, convergence checks, step time, total time, incrementation and other important information written for each iteration and increment of each step c) can be opened with wordpad
.dat	<ul style="list-style-type: none"> a) Print output file written by the analysis (you can use *Node Print, *El Print in the .inp file to write nodal and elemental outputs in a tabular format in the .dat file) b) contains any error or warning messages with other important information written for each iteration and increment of each step c) can be opened with wordpad
.fil	<ul style="list-style-type: none"> a) Results file written by the analysis b) You can use *Node File, *El File in the .inp file to write nodal and elemental outputs to the .fil file of one analysis, which can be used in another analysis. Ex: sequentially coupled thermo-mechanical analysis c) can’t be opened with wordpad or ABAQUS viewer

Introduction: some basics

Command	Purpose
module load abaqus	to invoke abaqus modules in EOS
abaqus cae	to launch abaqus
abaqus cae -mesa	to launch abaqus (if there is graphics problem with above command)
abaqus job=jobname.inp	to run the input file "jobname.inp"
module load intel-compilers	to load fortran compilers in EOS
abaqus job=jobname.inp user=fortranname.f	to run an input file "jobname.inp" along with user subroutine "fortranname.f"
abaqus fetch job=jobname.inp	to copy an input file from abaqus example problems manual to your present working directory
man abaqus	to get a detailed description of various commands of abaqus on EOS
abaqus help	some more commands



Introduction: some basics

Before going into examples you should know the following things

- **Units:**

Abaqus has no units built into it (except for rotational DOF (radians) and other angle measures(degrees)). Therefore, the units chosen must be self-consistent >> Refer: 1.2.2 Analysis user's manual

- **DOF:** Primary variables

a) Ex: Temperature(for heat transfer analysis); Displacements-translational & rotational(for mechanical analysis); Temperature + displacements (for thermomechanical analysis)

b) Displacements or other degrees of freedom are calculated at the nodes of the element. At any other point in the element, the displacements are obtained by interpolating from the nodal displacements.

- **Stress and strain measures:**

a) Stress is always reported as >> Cauchy or true stress

b) Shear strain is always reported as >> engineering shear strain

c) True strain is not that useful and therefore ABAQUS has different strain measures (Integrated strain, Green's Strain, Nominal Strain, Logarithmic strain>> Refer: 1.2.2 Analysis user's manual)

d) By default Stress and strains are calculated at integration points. If specified ABAQUS can interpolate these values to obtain at nodes or centroid of the element

- **Time:**

a) Step time>> measured from the beginning of each step

b) Total time>> starts at zero and is the total accumulated time over all the steps(except Linear perturbation)

Introduction: some basics

■ Incrementation:

Each step in an Abaqus analysis is divided into multiple increments. And 2 choices for incrementation.

a) Automatic Incrementation>> Just define the step and specify certain tolerances or error measures, Abaqus then automatically selects the increment size as it develops the response in the step.

b) Fixed incrementation:>> Increment size is specified by the user. If you have a good “feel” for the convergence behavior of the problem.

c) Automatic incrementation is recommended for most cases.

■ Types of Iterations:

ABAQUS attempts to perform multiple iterations for each increment until convergence is obtained

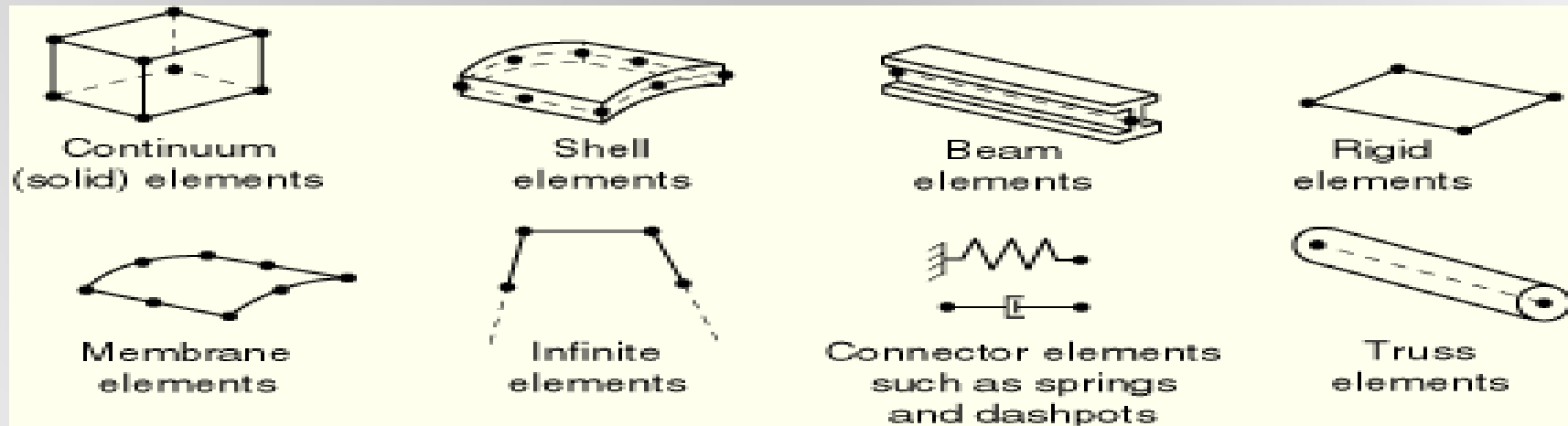
a) Equilibrium iterations >> the solution varies smoothly;

b) Severe discontinuity iterations (SDIs)>> abrupt changes in stiffness occur.

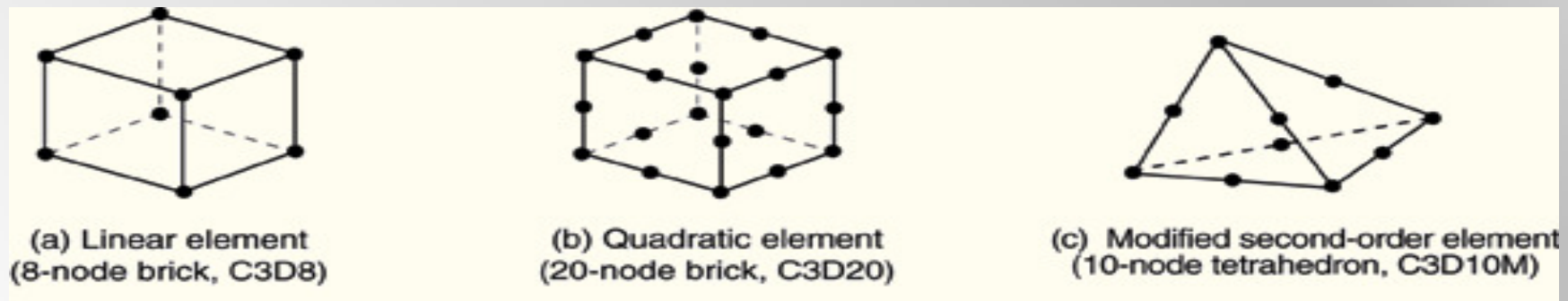
c) These can be seen in “.sta” file

Quantity	Abaqus/Standard	Abaqus/Explicit
Element library	Offers an extensive element library.	Offers an extensive library of elements well suited for explicit analyses. The elements available are a subset of those available in Abaqus/Standard.
Analysis procedures	General and linear perturbation procedures are available.	General procedures are available.
Material models	Offers a wide range of material models.	Similar to those available in Abaqus/Standard; a notable difference is that failure material models are allowed.
Contact formulation	Has a robust capability for solving contact problems.	Has a robust contact functionality that readily solves even the most complex contact simulations.
Solution technique	Uses a stiffness-based solution technique that is unconditionally stable.	Uses an explicit integration solution technique that is conditionally stable.
Disk space and memory	Due to the large numbers of iterations possible in an increment, disk space and memory usage can be large.	Disk space and memory usage is typically much smaller than that for Abaqus/Standard.

Introduction: Element types

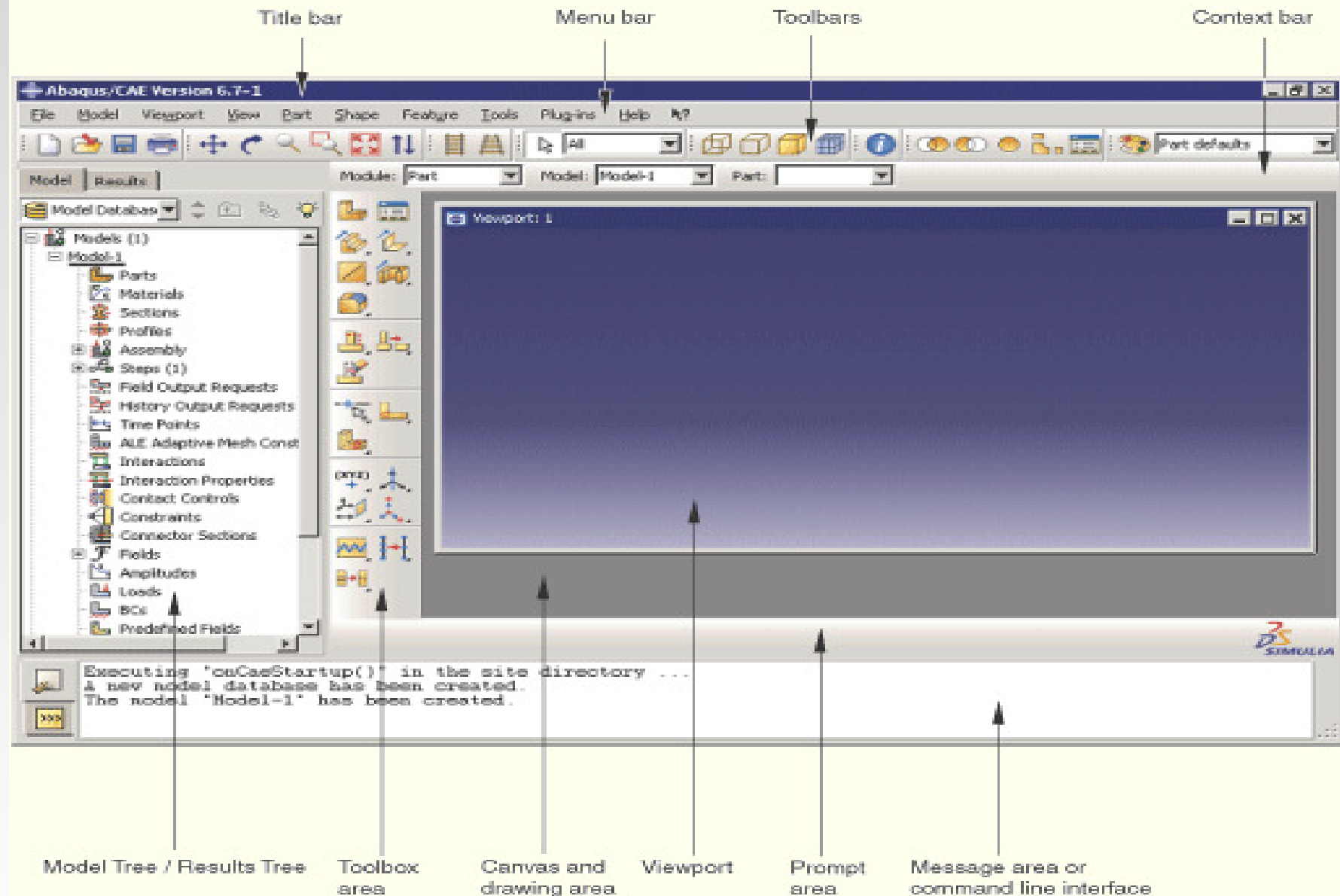


- a) Commonly used element families in a stress analysis.
- b) The first letter indicates to which family the element belongs (S4R>> shell element; C3D8>> continuum element)



- c) # of nodes in an element is clearly identified in its name. (S4R>> 4 node shell element ; C3D8>> 8 node brick element)

Introduction: ABAQUS CAE interface





Example 1: 3D Stress Analysis

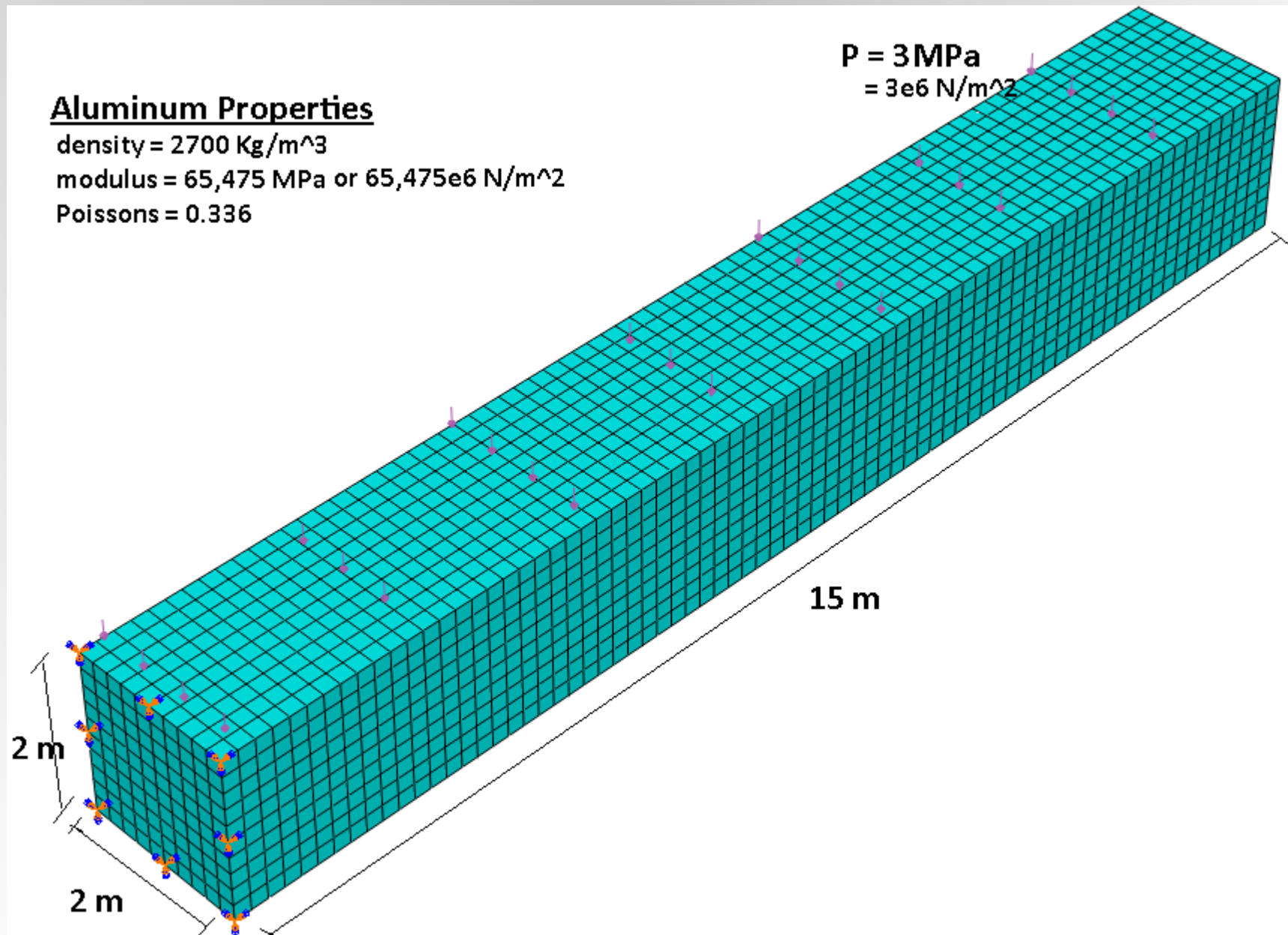
Problem description

Aluminum Properties

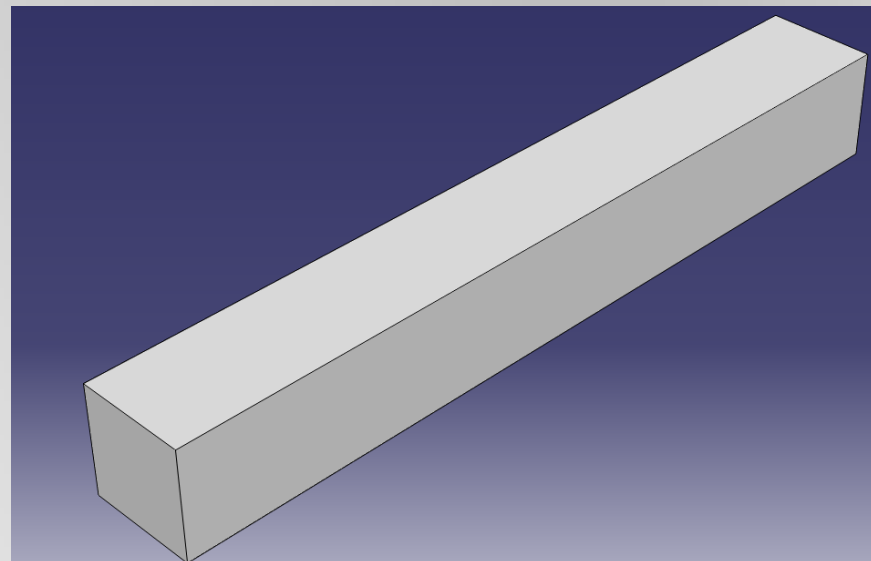
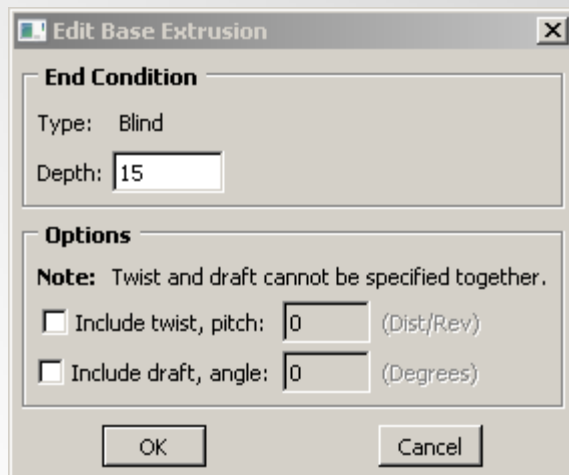
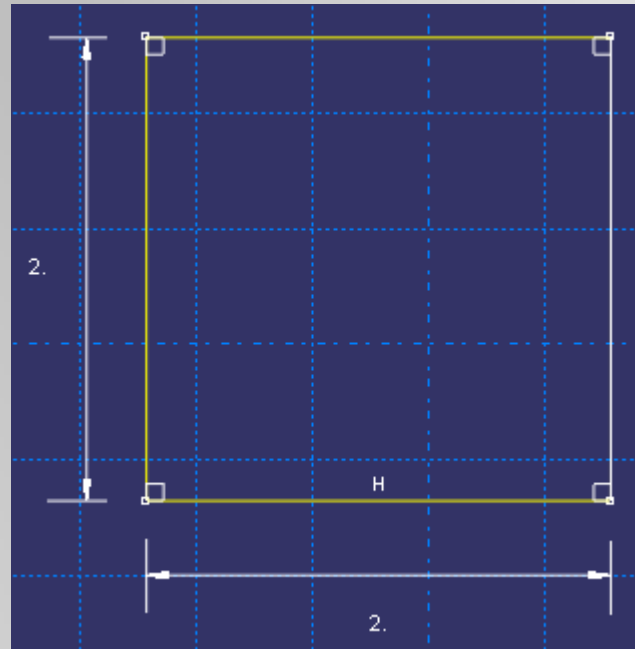
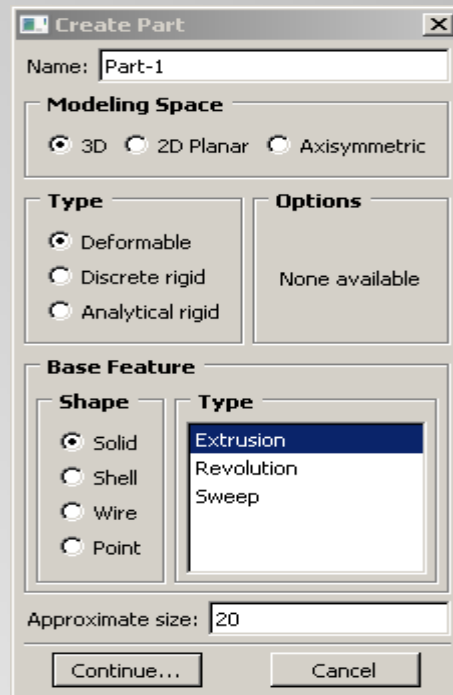
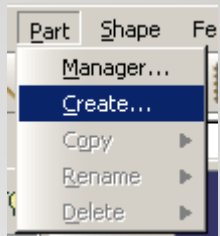
density = 2700 Kg/m^3

modulus = $65,475 \text{ MPa}$ or $65,475 \text{e}6 \text{ N/m}^2$

Poissons = 0.336



Part module



Property module

The screenshot displays the ANSYS Property module interface with four dialog boxes open:

- Edit Material:** Shows the material 'Aluminum' with the 'Elastic' behavior selected. The 'Elastic' properties are set to 'Isotropic' type, 'No temperature-dependent data', '0' field variables, and 'Long-term' moduli time scale. The 'Data' table shows a Young's Modulus of 65475e6 and a Poisson's Ratio of 0.336.
- Create Section:** Shows a new section named 'Section-1' with a 'Solid' category and 'Homogeneous' type.
- Edit Section:** Shows the 'Section-1' properties, including 'Solid, Homogeneous' type, 'Aluminum' material, and a 'Plane stress/strain thickness' of 1.
- Section Assignment Manager:** Shows the assignment of 'Section-1 (Solid, Homogeneous)' to 'Aluminum' material in the 'Region'.

The 'Section Assignment Manager' dialog box contains the following table:

Section Name (Type)	Material Name	Region
Section-1 (Solid, Homogeneous)	Aluminum	(Picked)

The section property provides any additional data required to define the geometry of the element and also identifies the associated material property definition.

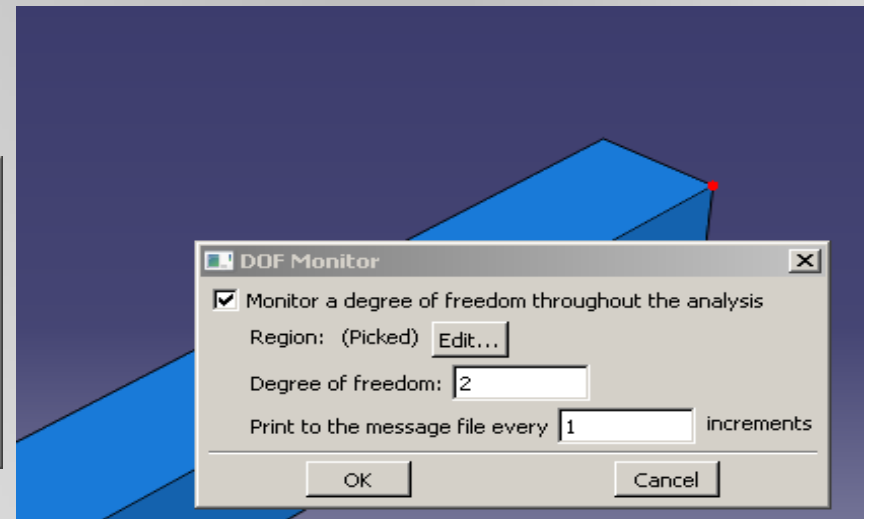
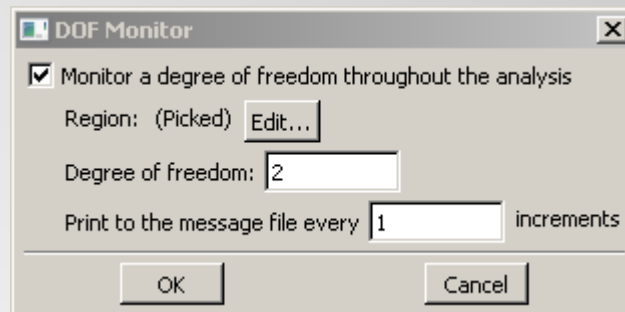
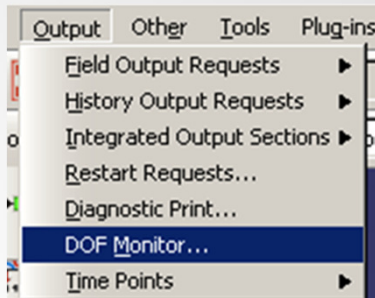
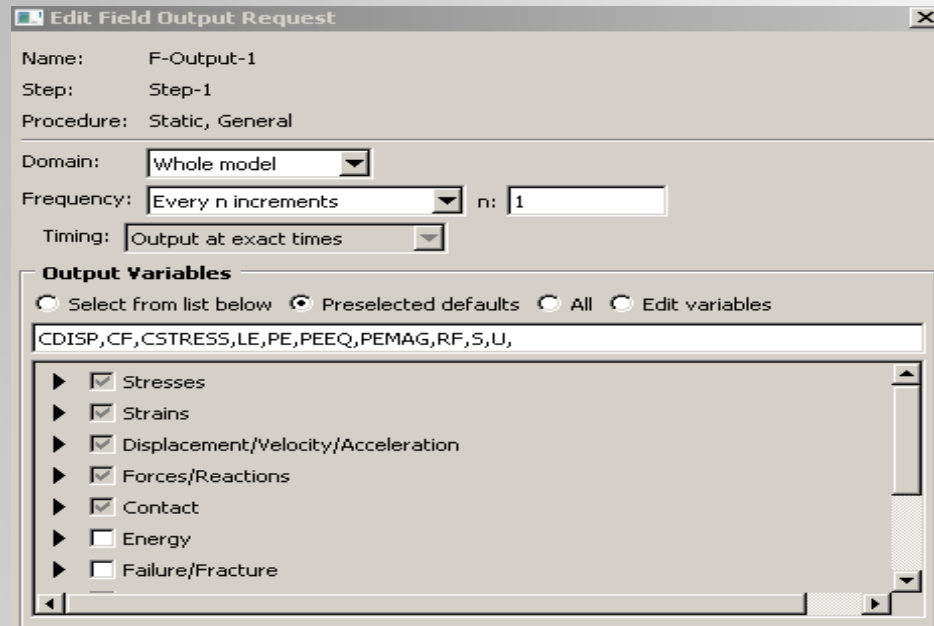
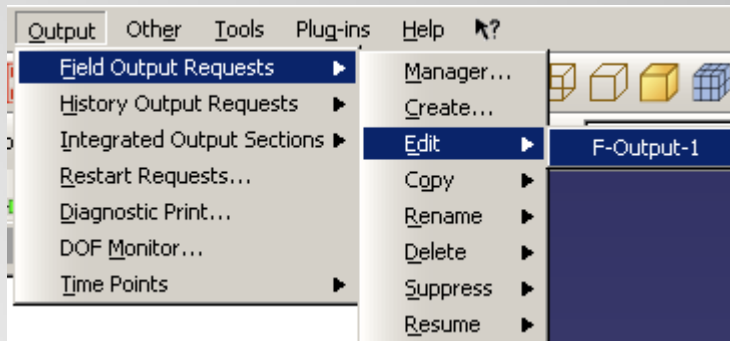
Assembly & Step module

The image displays the Abaqus Assembly & Step module interface with several key components:

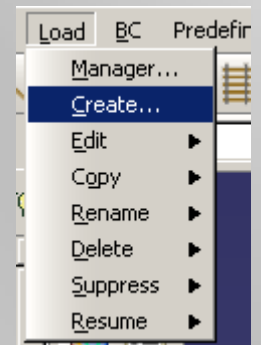
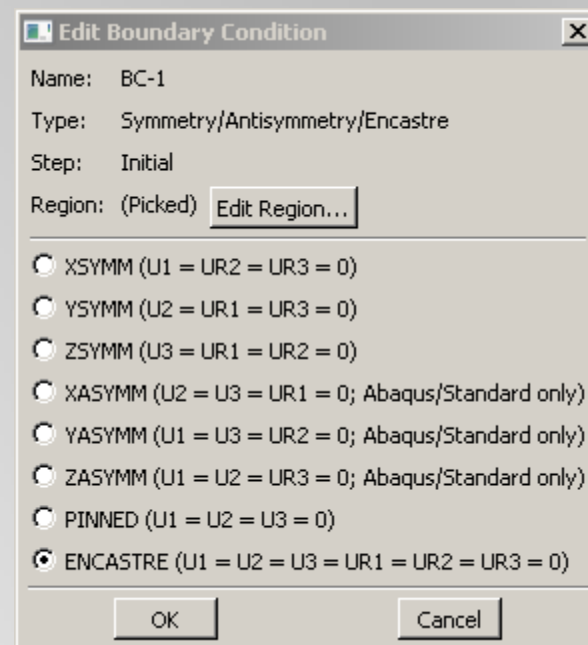
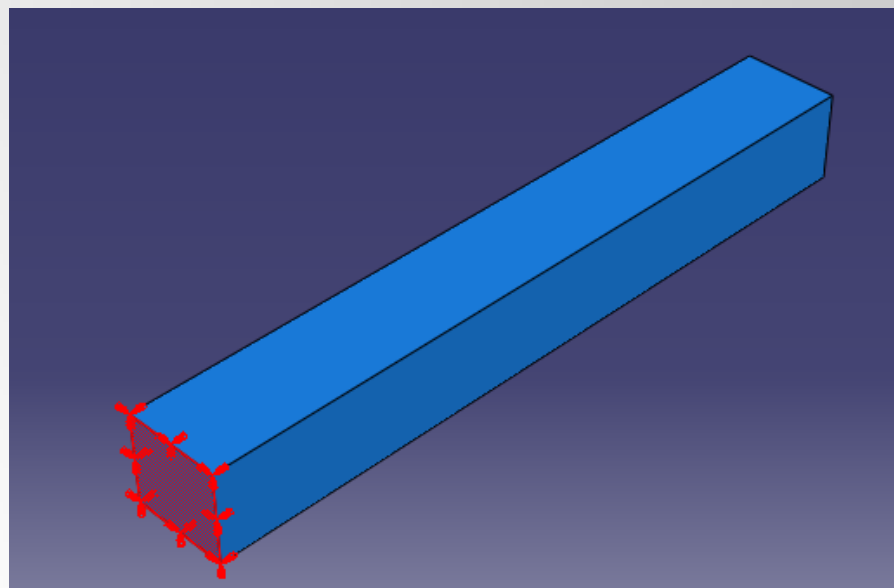
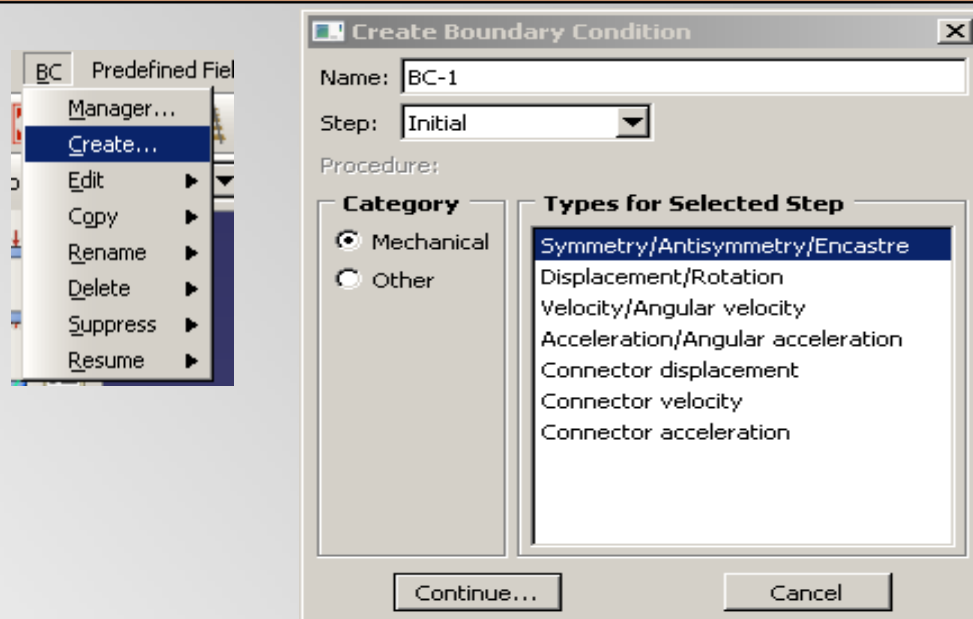
- Instance Constraint Filter:** A menu on the left with tabs for 'Instance', 'Constraint', and 'Filter'. The 'Instance' tab is active, showing a 'Create...' button and a list of options: Linear Pattern, Radial Pattern, Translate, Translate To, Rotate, Replace, Convert Constraints, and Merge/Cut...
- Create Instance Dialog:** A dialog box with a 'Parts' list containing 'Part-1'. Under 'Instance Type', 'Dependent (mesh on part)' is selected. A note states: 'To change a Dependent instance's mesh, you must edit its part's mesh.' There is also an 'Auto-offset from other instances' checkbox and 'OK', 'Apply', and 'Cancel' buttons.
- Step Manager Menu:** A context menu with 'Step' and 'Output' tabs. The 'Step' tab is active, showing options: Manager..., Create..., Edit, Replace, Rename, Delete, and Nlgeom...
- Create Step Dialog:** A dialog box for creating a new step. 'Name' is 'Step-1'. 'Insert new step after' is 'Initial'. 'Procedure type' is set to 'General'. A list of procedure types is shown, with 'Static, General' selected. 'Continue...' and 'Cancel' buttons are at the bottom.
- Edit Step Dialog (Left):** A dialog box for editing 'Step-1'. 'Type' is 'Static, General'. It has tabs for 'Basic', 'Incrementation', and 'Other'. The 'Basic' tab is active, showing a 'Description' field, 'Time period' set to '10', 'Nlgeom' set to 'Off', and 'Automatic stabilization' set to 'None'. There is also an 'Include adiabatic heating effects' checkbox.
- Edit Step Dialog (Right):** Another 'Edit Step' dialog for 'Step-1'. 'Type' is 'Static, General'. It has tabs for 'Basic', 'Incrementation', and 'Other'. The 'Incrementation' tab is active, showing 'Type' set to 'Automatic', 'Maximum number of increments' set to '100', and an 'Increment size' table.

	Initial	Minimum	Maximum
Increment size:	1	0.0001	10

Step module: Field Output & DOF monitor



Load module : defining BC & Loads



Load module: defining BC & Load

Create Load

Name: Load-1

Step: Step-1

Procedure: Static, General

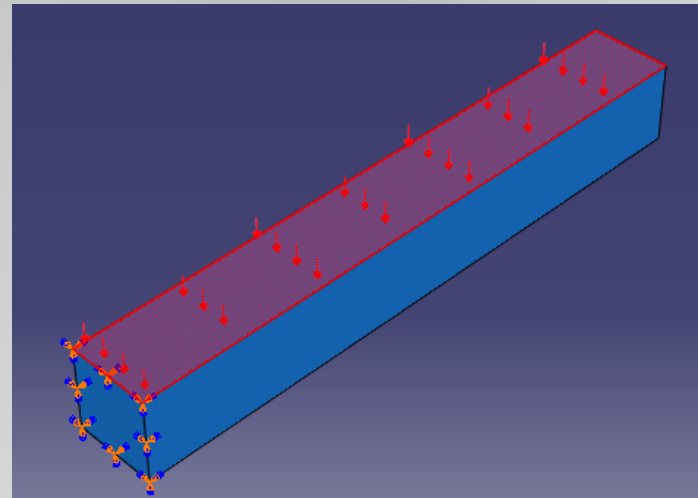
Category

- ☒ Mechanical
- ☐ Thermal
- ☐ Acoustic
- ☐ Fluid
- ☐ Electrical
- ☐ Mass diffusion
- ☐ Other

Types for Selected Step

- Concentrated force
- Moment
- Pressure**
- Shell edge load
- Surface traction
- Pipe pressure
- Body force
- Line load
- Gravity
- Bolt load

Continue... Cancel



Edit Load

Name: Load-1

Type: Pressure

Step: Step-1 (Static, General)

Region: (Picked) Edit Region...

Distribution: Uniform Create...

Magnitude: 3e6

Amplitude: (Ramp) Create...

OK Cancel

Boundary Condition Manager

Name	Initial	Step-1
✓ BC-1	Created	Propagated

Edit... Move Left Move Right Activate Deactivate

Step procedure:

Boundary condition type: Symmetry/Antisymmetry/Encastre

Boundary condition status: Created in this step

Create... Copy... Rename... Delete... Dismiss

Load Manager

Name	Step-1
✓ Load-1	Created

Edit... Move Left Move Right Activate Deactivate

Step procedure: Static, General

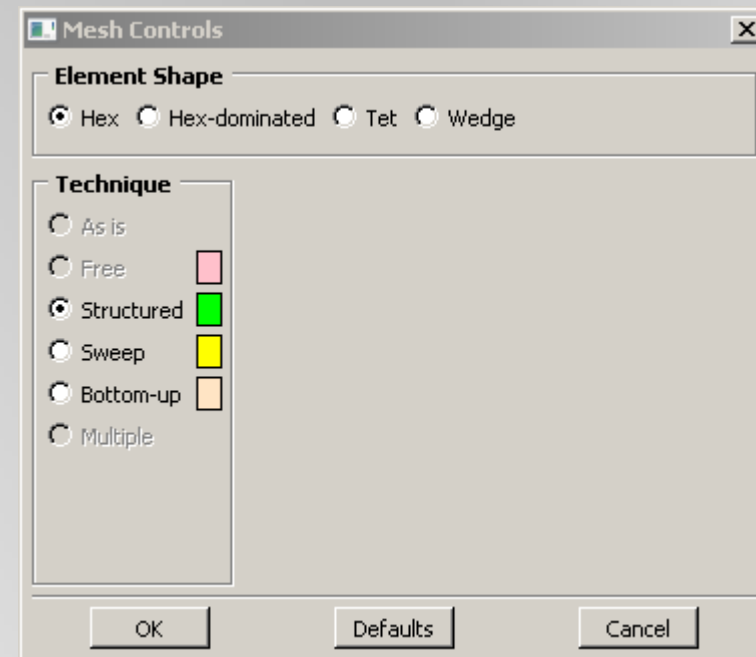
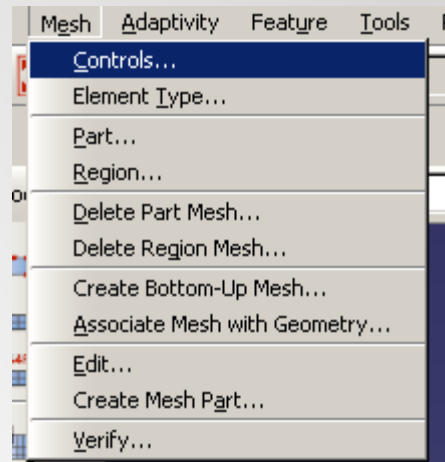
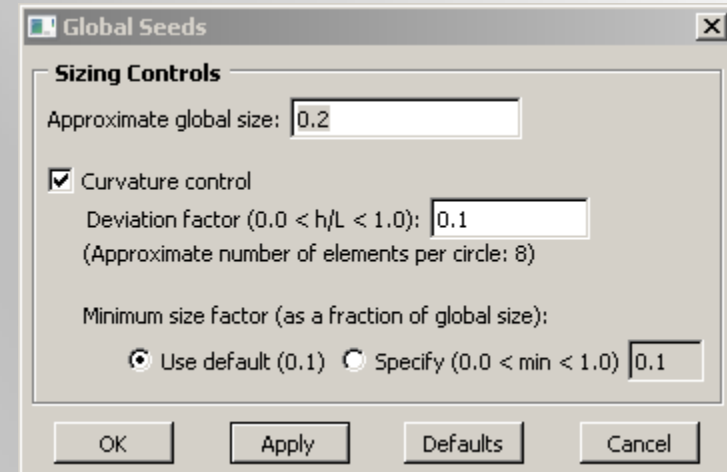
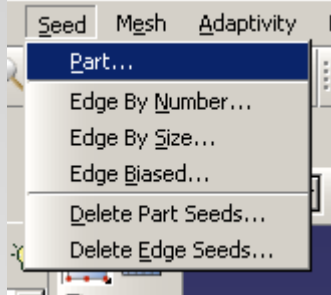
Load type: Pressure

Load status: Created in this step

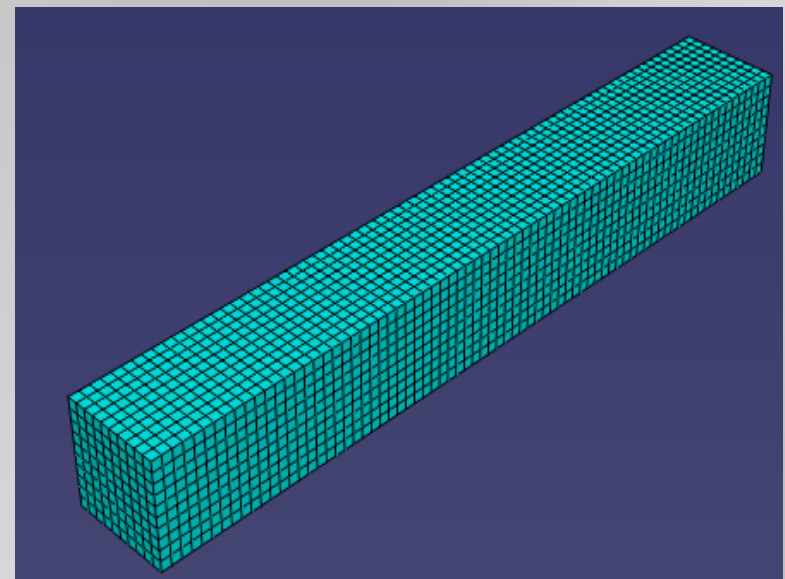
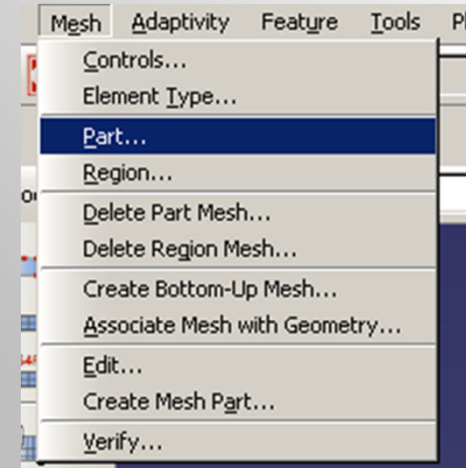
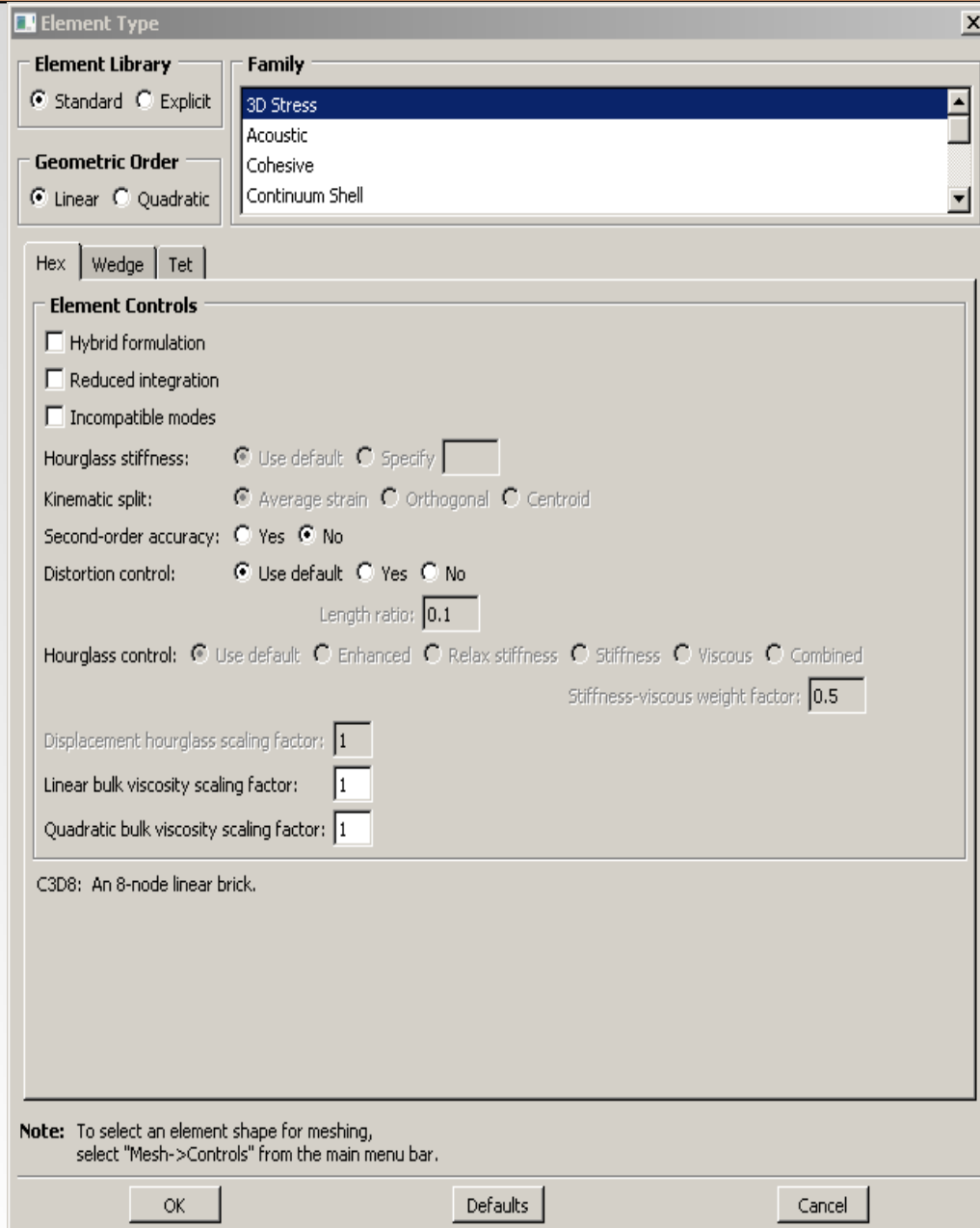
Create... Copy... Rename... Delete... Dismiss

Mesh Module

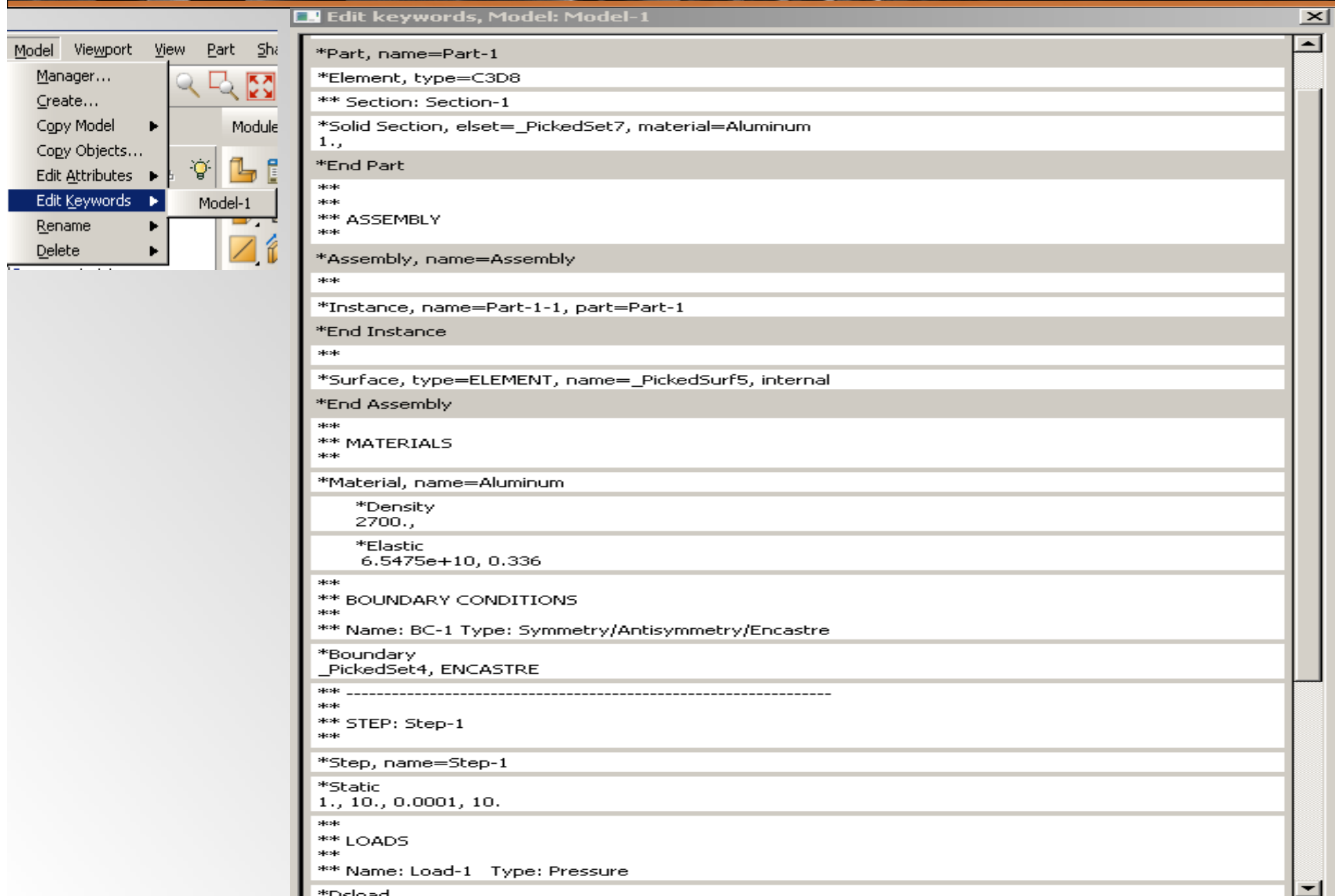
Module: **Mesh** Model: **Model-1** Object: ☐ Assembly ☒ Part:



Mesh Module



Job Module



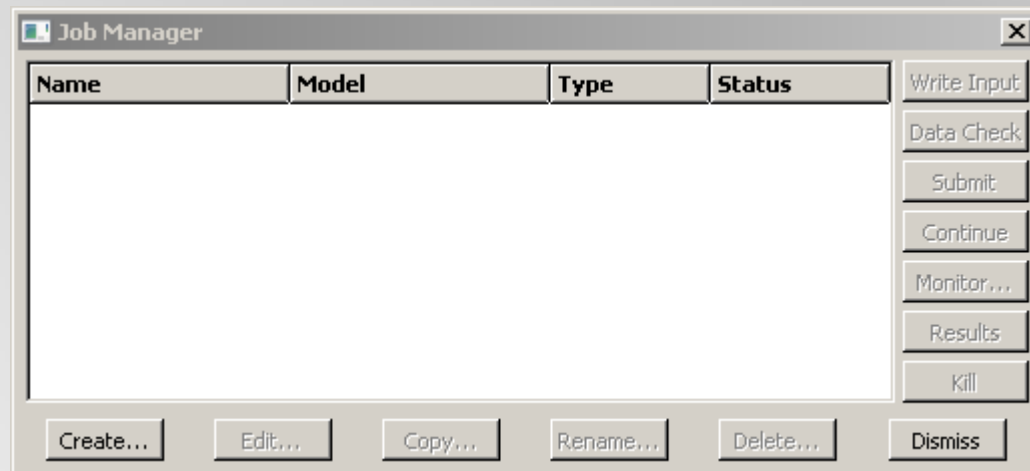
Edit keywords, Model: Model-1

```

*Part, name=Part-1
*Element, type=C3D8
** Section: Section-1
*Solid Section, elset=_PickedSet7, material=Aluminum
1.,
*End Part
***
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=Part-1-1, part=Part-1
*End Instance
**
*Surface, type=ELEMENT, name=_PickedSurf5, internal
*End Assembly
**
** MATERIALS
**
*Material, name=Aluminum
    *Density
    2700.,
    *Elastic
    6.5475e+10, 0.336
**
** BOUNDARY CONDITIONS
**
** Name: BC-1 Type: Symmetry/Antisymmetry/Encastre
*Boundary
_PickedSet4, ENCASTRE
** -----
**
** STEP: Step-1
**
*Step, name=Step-1
*Static
1., 10., 0.0001, 10.
**
** LOADS
**
** Name: Load-1 Type: Pressure
*Load

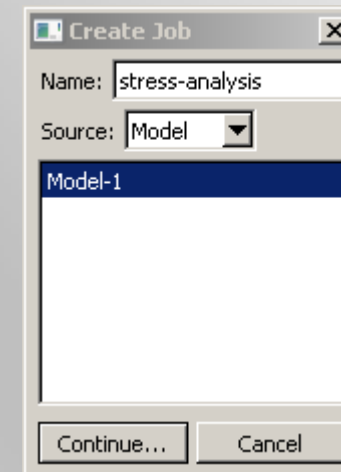
```


Job Module



The Job Manager dialog box features a table with four columns: Name, Model, Type, and Status. The table is currently empty. To the right of the table is a vertical stack of buttons: Write Input, Data Check, Submit, Continue, Monitor..., Results, and Kill. At the bottom of the dialog is a horizontal row of buttons: Create..., Edit..., Copy..., Rename..., Delete..., and Dismiss.

Name	Model	Type	Status
------	-------	------	--------

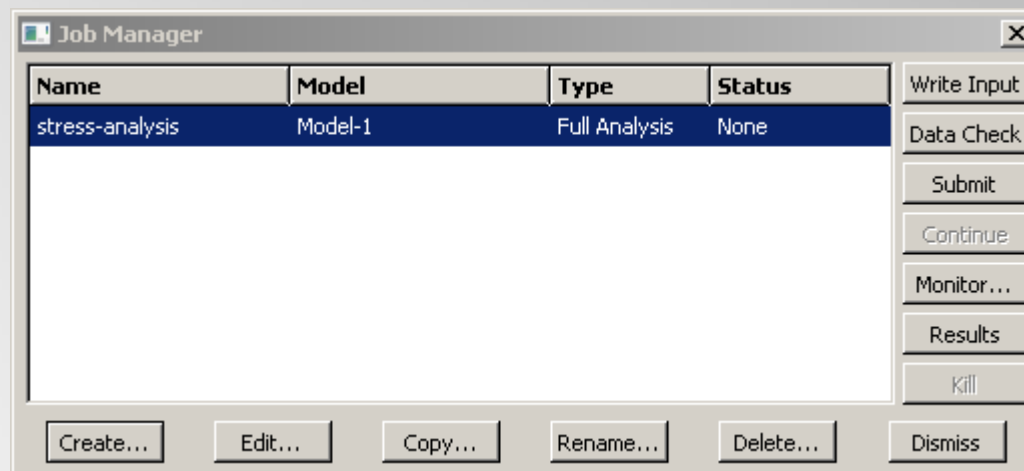


The Create Job dialog box contains a Name field with the text "stress-analysis". Below it is a Source dropdown menu currently set to "Model". Under the dropdown is a list box containing "Model-1". At the bottom are "Continue..." and "Cancel" buttons.

Name: stress-analysis

Source: Model

Model-1



The Job Manager dialog box now displays one job in the table. The table has columns: Name, Model, Type, and Status. The first row is highlighted in blue and contains the text "stress-analysis", "Model-1", "Full Analysis", and "None". The buttons and layout are the same as in the previous image.

Name	Model	Type	Status
stress-analysis	Model-1	Full Analysis	None

status of job : check ".sta" and ".msg" files for error messages

```
stress-analysis.sta - WordPad
File Edit View Insert Format Help

Abaqus/Standard Version 6.7-1          DATE 22-Jan-2012 TIME 14:17:10
SUMMARY OF JOB INFORMATION:
STEP  INC ATT SEVERE EQUIL TOTAL    TOTAL    STEP    INC OF
DOF    IF
DISCON ITERS ITERS  TIME/    TIME/LPF  TIME/LPF
MONITOR RIKS
ITERS          FREQ
1      1      1      0      1      1 1.00    1.00    1.000
1      2      1      0      1      1 2.00    2.00    1.000
1      3      1      0      1      1 3.50    3.50    1.500
1      4      1      0      1      1 5.75    5.75    2.250
1      5      1      0      1      1 9.13    9.13    3.375
1      6      1      0      1      1 10.0    10.0    0.8750

THE ANALYSIS HAS COMPLETED SUCCESSFULLY

For Help, press F1
```

```
stress-analysis.msg - WordPad
File Edit View Insert Format Help

THE ANALYSIS HAS BEEN COMPLETED

ANALYSIS SUMMARY:
TOTAL OF      6 INCREMENTS
              0 CUTBACKS IN AUTOMATIC INCREMENTATION
              6 ITERATIONS INCLUDING CONTACT ITERATIONS IF

PRESENT
              6 PASSES THROUGH THE EQUATION SOLVER OF WHICH
              1 INVOLVE MATRIX DECOMPOSITION, INCLUDING
              0 DECOMPOSITION(S) OF THE MASS MATRIX
              1 REORDERING OF EQUATIONS TO MINIMIZE WAVEFRONT
              0 ADDITIONAL RESIDUAL EVALUATIONS FOR LINE

SEARCHES
              0 ADDITIONAL OPERATOR EVALUATIONS FOR LINE

SEARCHES
              0 WARNING MESSAGES DURING USER INPUT PROCESSING
              0 WARNING MESSAGES DURING ANALYSIS
              0 ANALYSIS WARNINGS ARE NUMERICAL PROBLEM

MESSAGES
              0 ANALYSIS WARNINGS ARE NEGATIVE EIGENVALUE

MESSAGES
              0 ERROR MESSAGES

JOB TIME SUMMARY
USER TIME (SEC)      = 12.500
SYSTEM TIME (SEC)    = 2.0000
TOTAL CPU TIME (SEC) = 14.500
WALLCLOCK TIME (SEC) = 15

For Help, press F1
```

Visualization Module

Viewport: 1

Plot Animate Report Options Tools Plug-ins Help

Query...
Coordinate System
Color Code...
Display Group
XY Data
Create Field Output
Path
Spectrum
View Cut
Job Diagnostics...
Movie
Customize...

Module: Visualization ODB: new-sa.odb

Visualization defaults

Job Diagnostics

Job History

- Job
 - Step 1
 - Increment 1
 - Attempt 1
 - Iteration 1
 - Increment 2
 - Increment 3
 - Increment 4
 - Increment 5
 - Increment 6

Equations
☒ Field
☐ Constraint

Variables
 Displacement

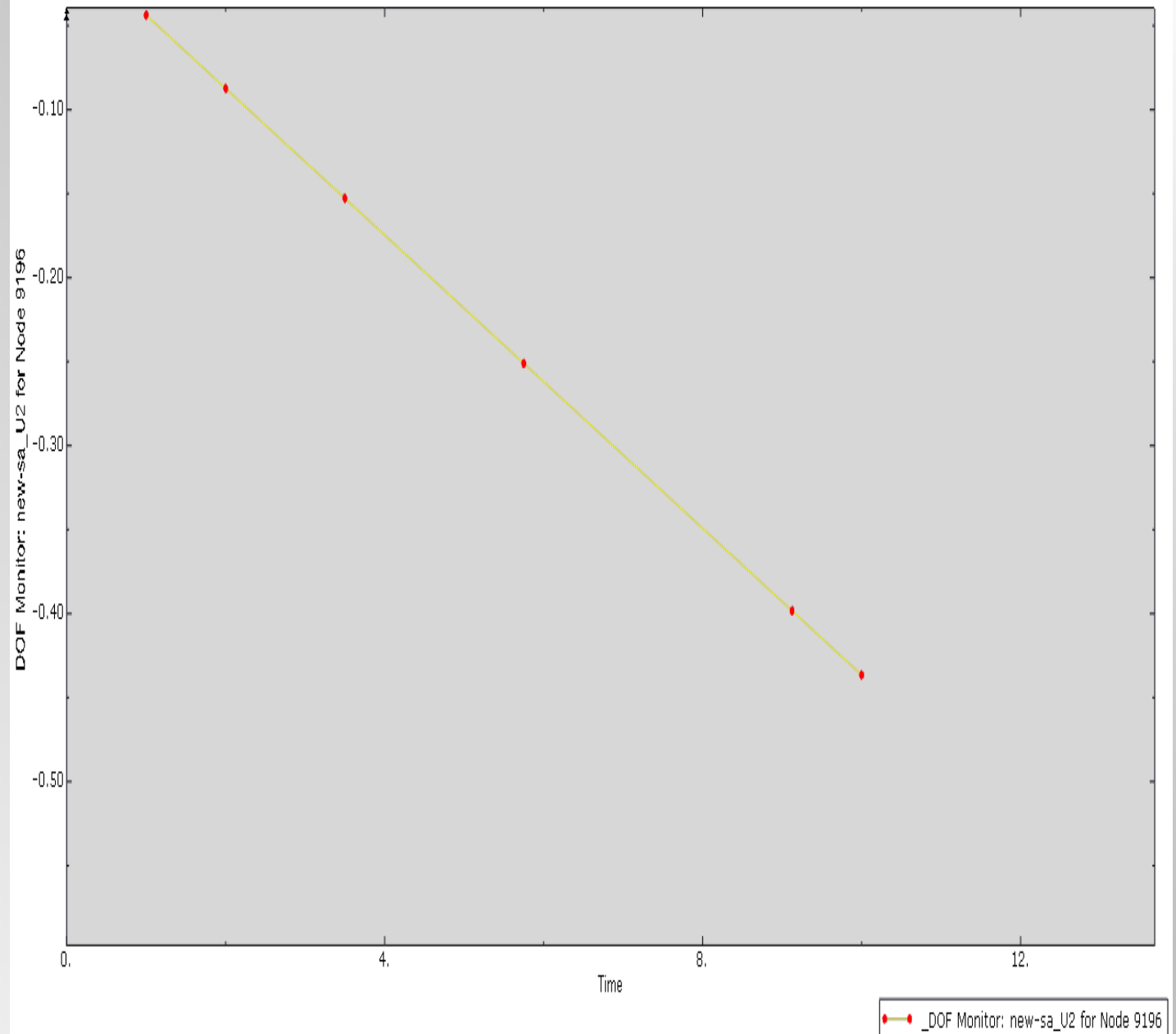
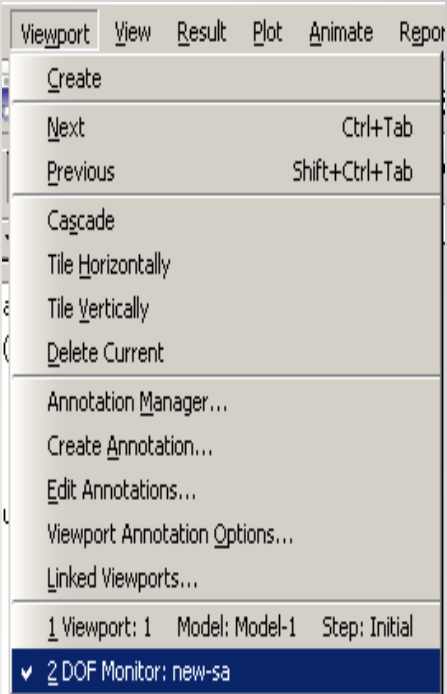
Details
 The force equilibrium response was linear.
 Average force: 24320.1
 Time average force: 24320.1

Description	Value	DC
Max force residual	1.48463e-06	2
Max displacement increment	-0.0436502	2
Max displacement correction	-0.0436502	2

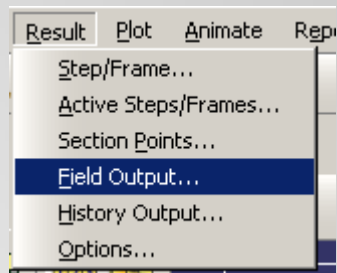
☐ Highlight selection in viewport

Dismiss

Visualization Module: DOF monitor(2nd viewport)



Visualization Module



Field Output

Step/Frame
Step: 1, Step-1
Frame: 6 [Step/Frame...](#)

Primary Variable Deformed Variable Symbol Variable Status Variable

Output Variable
☐ List only variables with results: [...](#)

Name	Description (* indicates complex)
AC YIELD	Active yield flag at integration points
CF	Point loads at nodes
E	Strain components at integration points
PE	Plastic strain components at integration points
PEEQ	Equivalent plastic strain at integration points
PEMAG	Magnitude of plastic strain at integration points
RF	Reaction force at nodes
S	Stress components at integration points
U	Spatial displacement at nodes

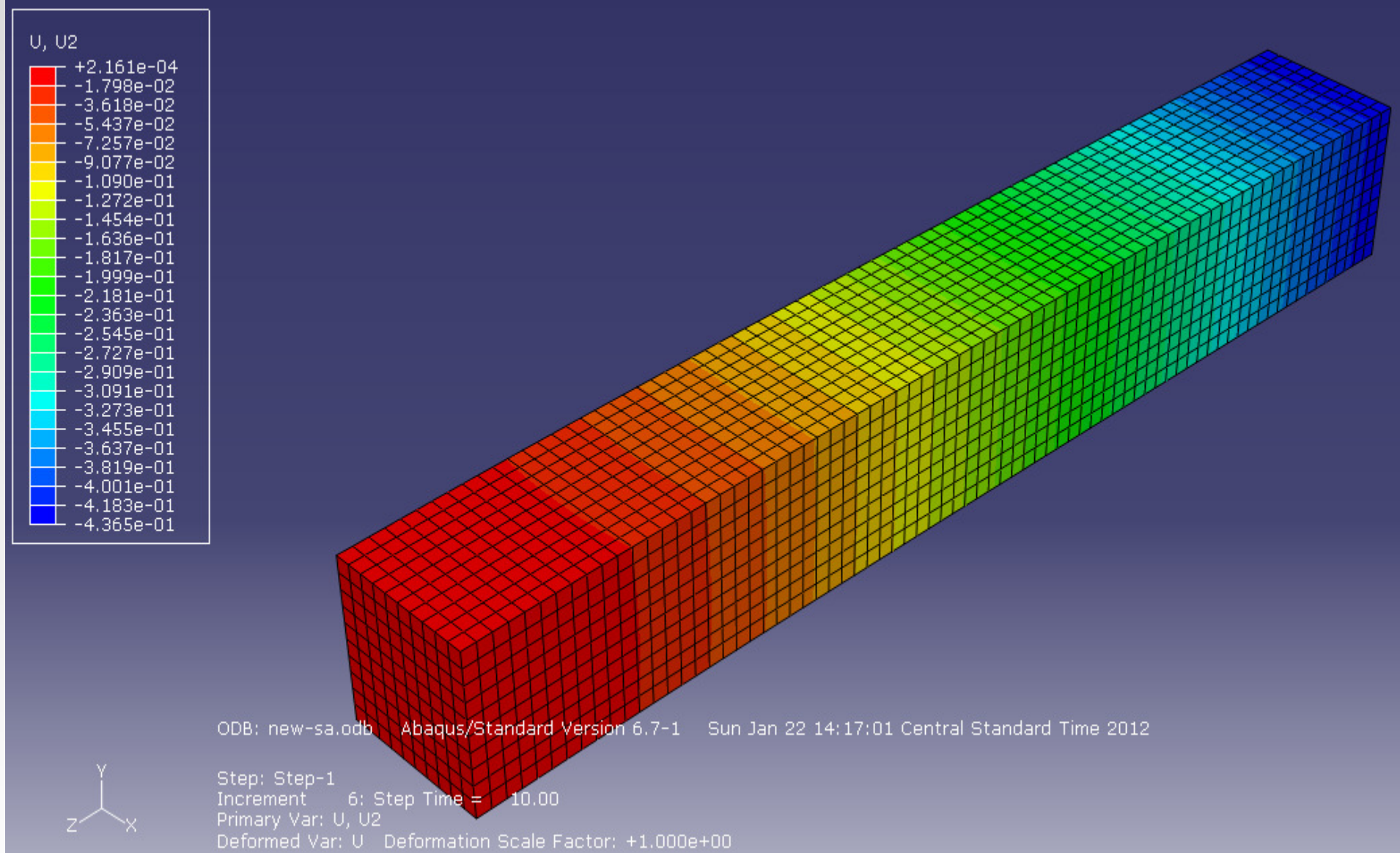
Invariant
Magnitude

Component
U1
U2
U3

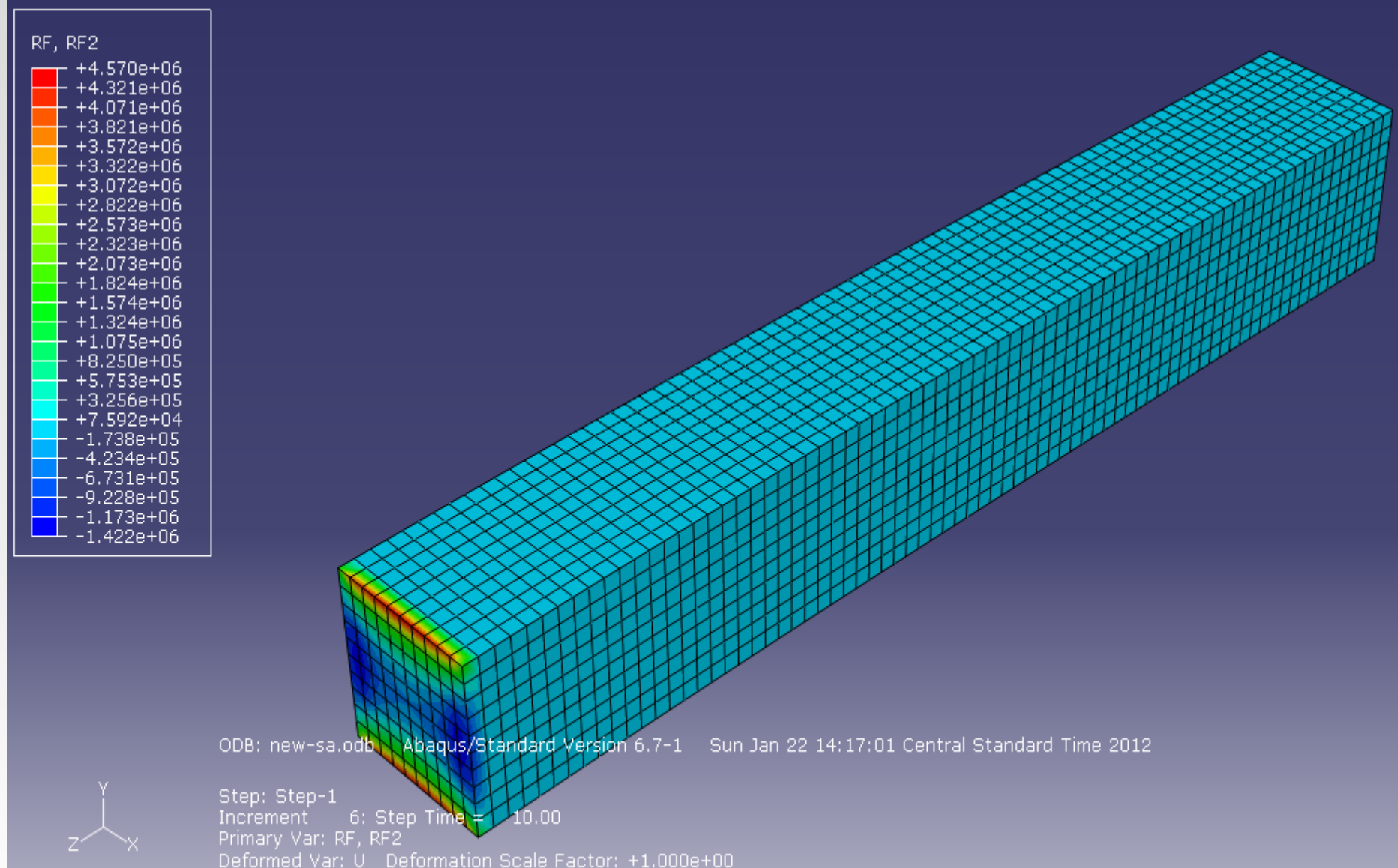
[Section Points...](#)

OK Apply Cancel

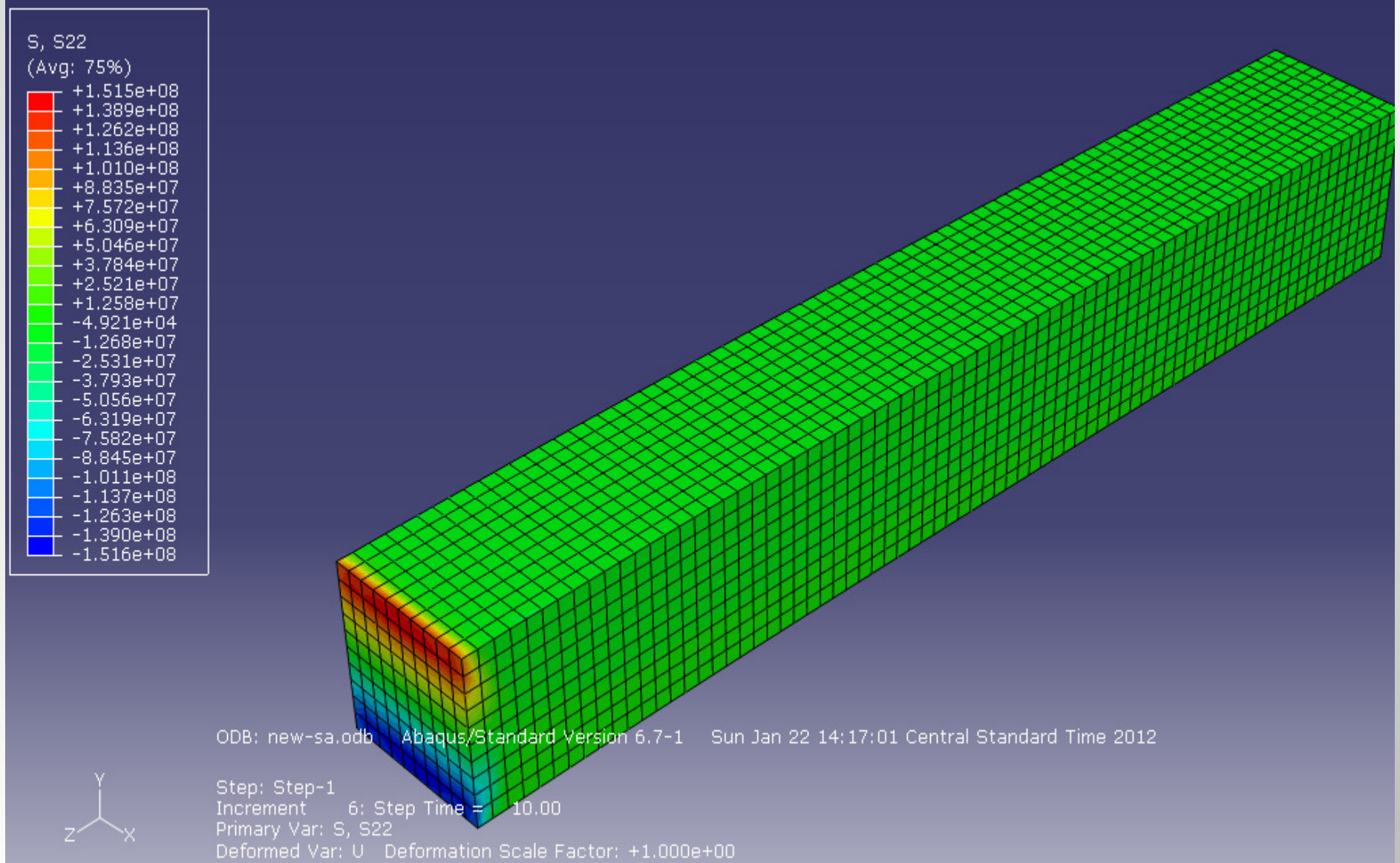
Visualization Module: Vertical displacement



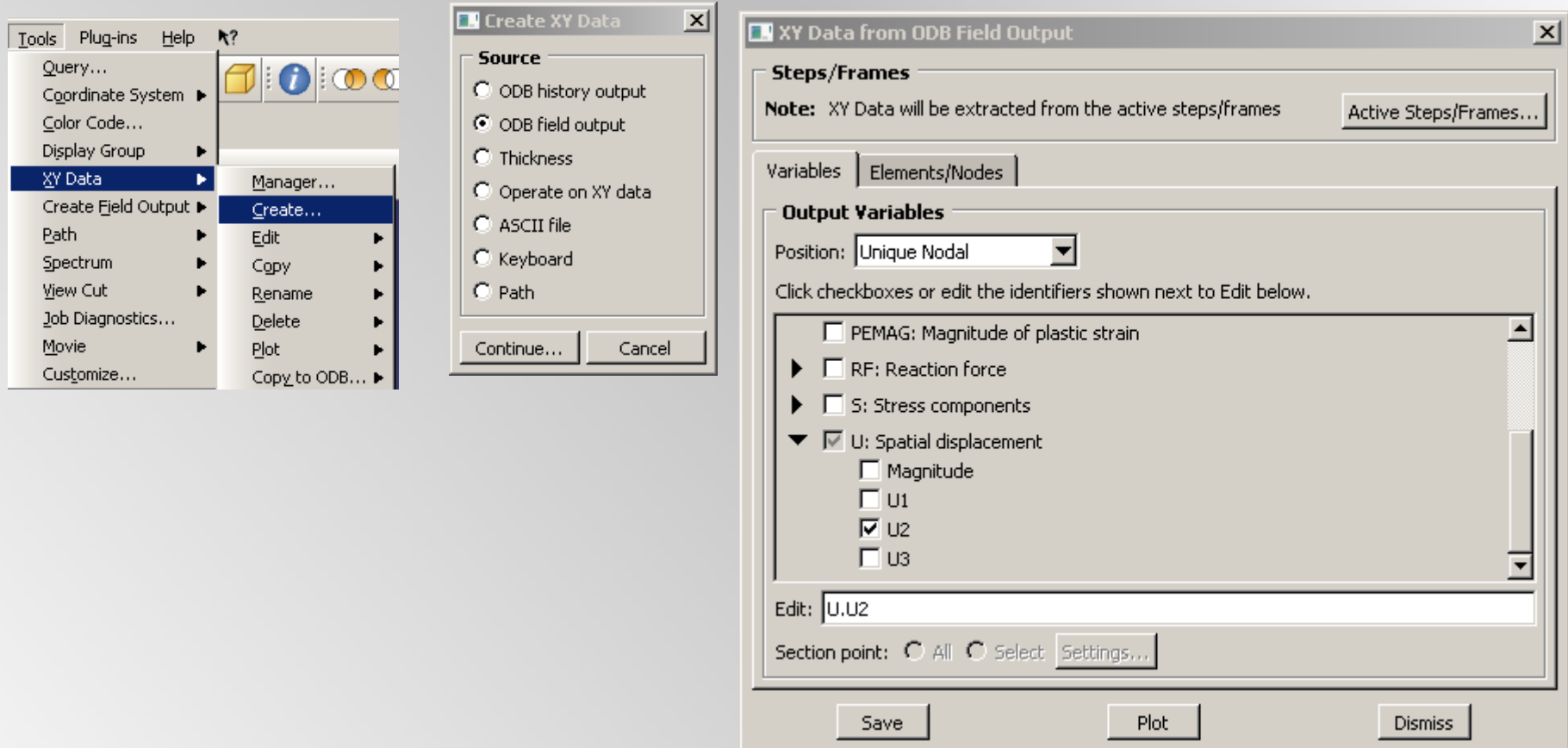
Visualization Module: Reaction forces



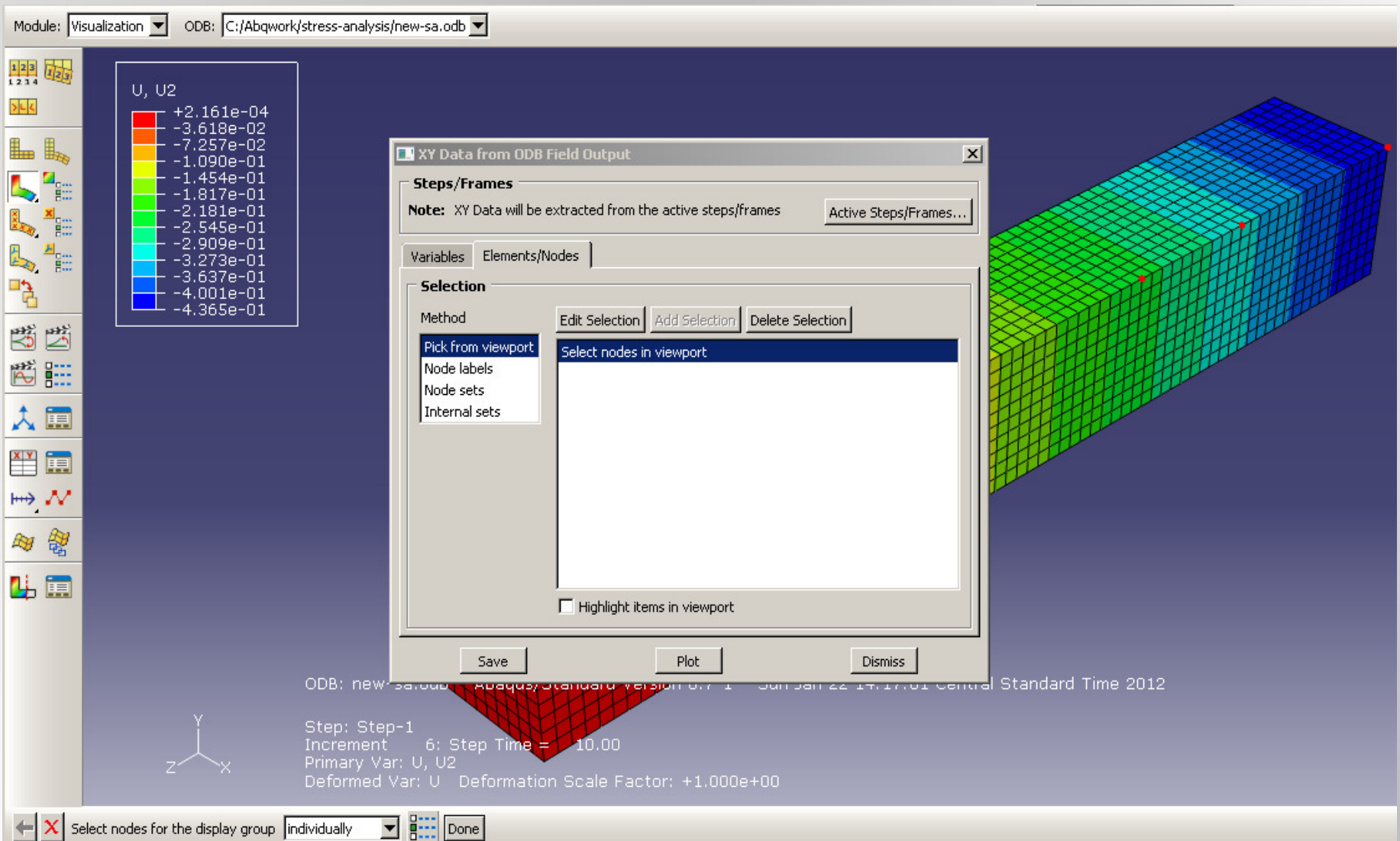
Visualization Module: Stress



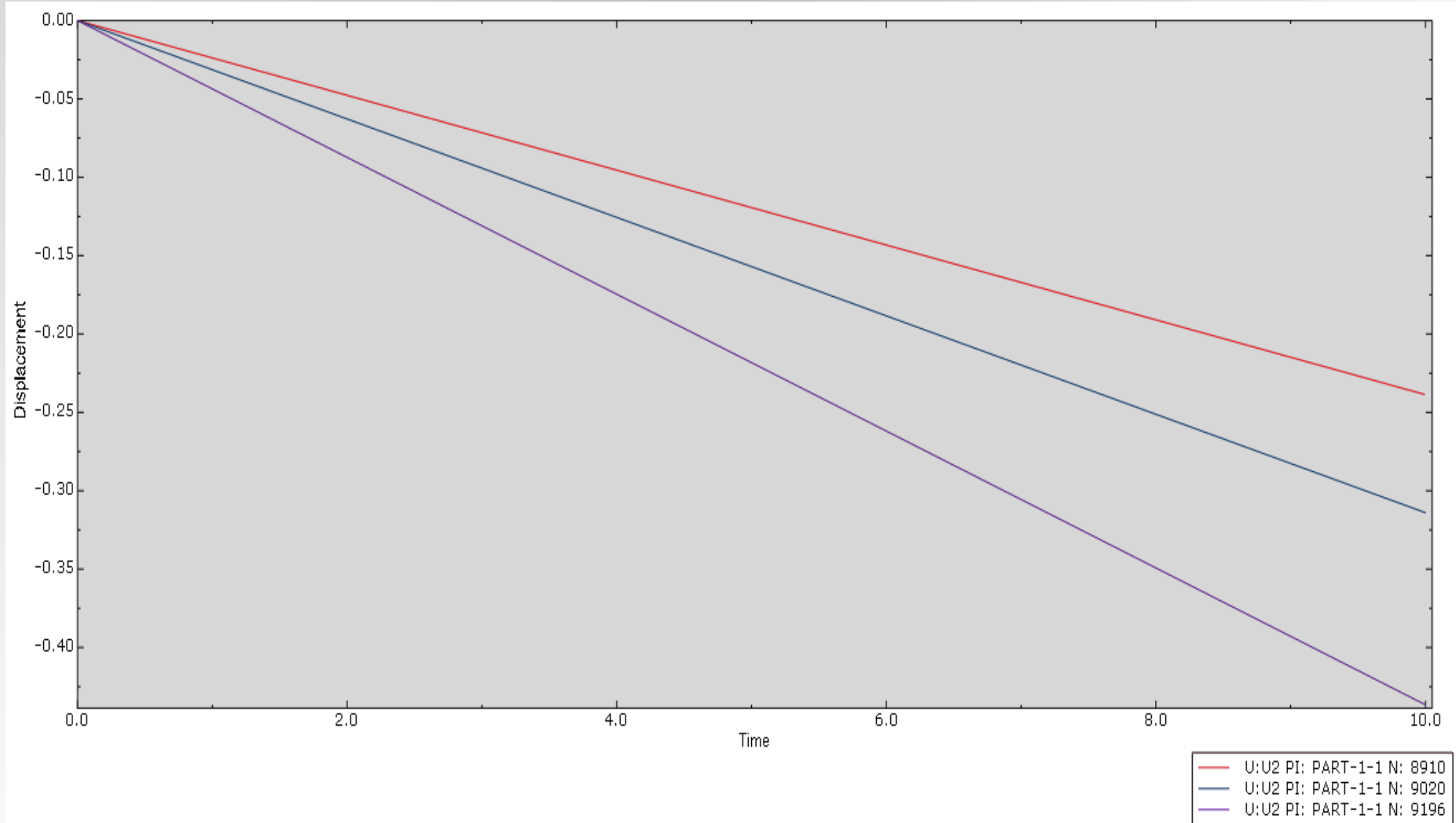
Visualization Module: XY Plot



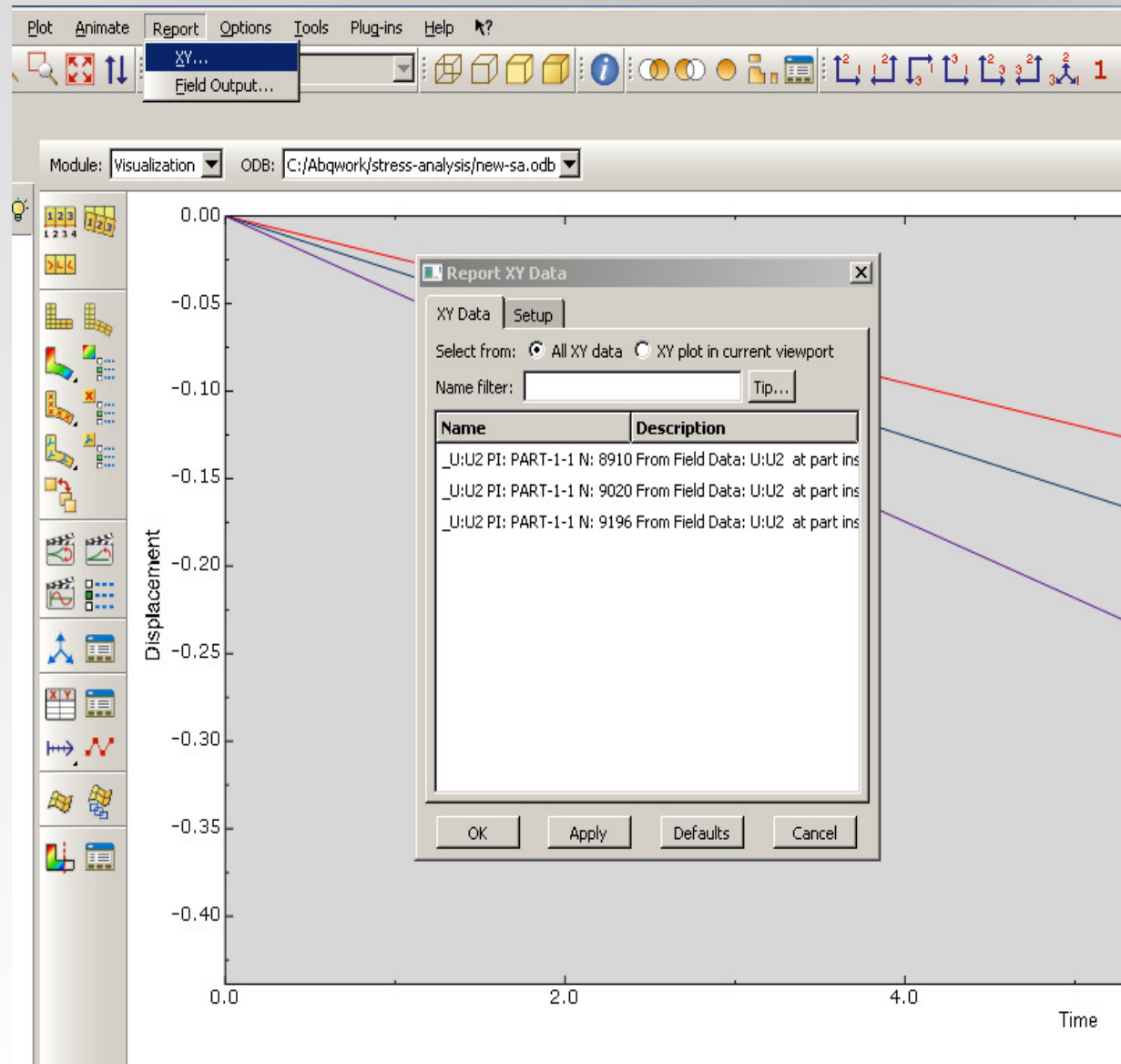
Visualization Module: XY Plot



Visualization Module: XY Plot



Visualization Module: XY data transfer to excel



The 'Report XY Data' dialog box is shown with the 'Setup' tab selected. The 'File' section has 'Name' set to 'abaqus.rpt' and 'Append to file' checked. The 'Output Format' section has 'Layout' set to 'Single table for all XY data', 'Interpolate between X values (if necessary)' unchecked, 'Separate table for each XY data' unchecked, 'Page width (characters)' set to 'No limit', 'Number of significant digits' set to 6, and 'Number format' set to 'Engineering'. The 'Data' section has 'Write' checked for 'XY data', 'Column totals' unchecked, and 'Column min/max' unchecked. The 'OK', 'Apply', 'Defaults', and 'Cancel' buttons are visible at the bottom.

File
Name: abaqus.rpt
Append to file: <input checked="" type="checkbox"/>

Output Format
Layout: <input checked="" type="radio"/> Single table for all XY data
<input type="checkbox"/> Interpolate between X values (if necessary)
<input type="radio"/> Separate table for each XY data
Page width (characters): <input checked="" type="radio"/> No limit <input type="radio"/> Specify: 80
Number of significant digits: 6
Number format: Engineering

Data
Write: <input checked="" type="checkbox"/> XY data <input type="checkbox"/> Column totals <input type="checkbox"/> Column min/max



Example 2: Transient Heat Transfer Analysis

Refer: Section 2.11.1 Abaqus Theory manual

Problem description

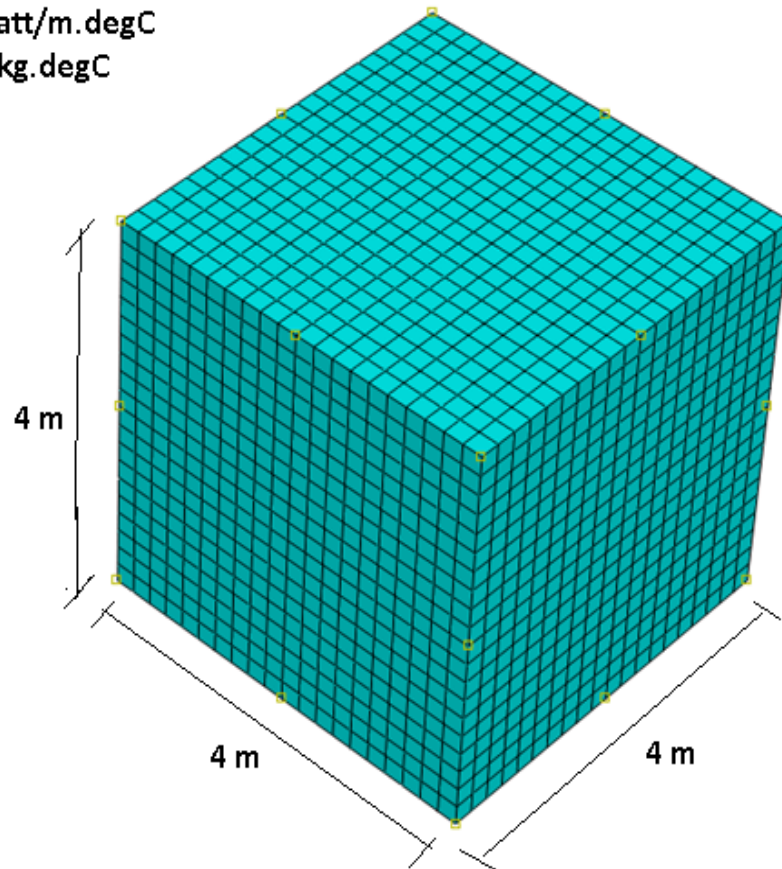
Aluminum Properties

density = 2700 Kg/m^3

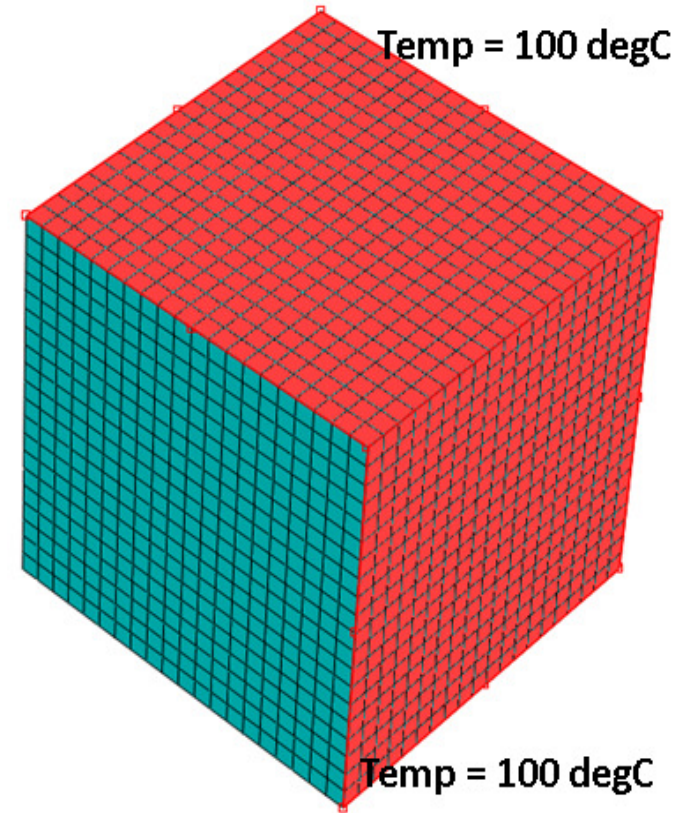
conductivity = 210 Watt/m.degC

specific heat = 900 J/kg.degC

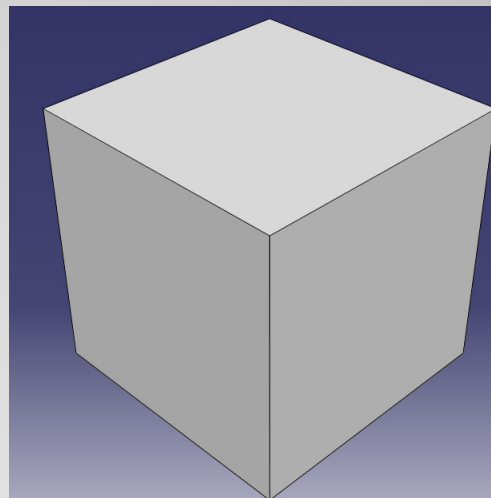
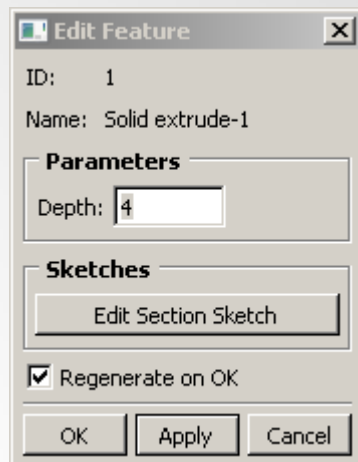
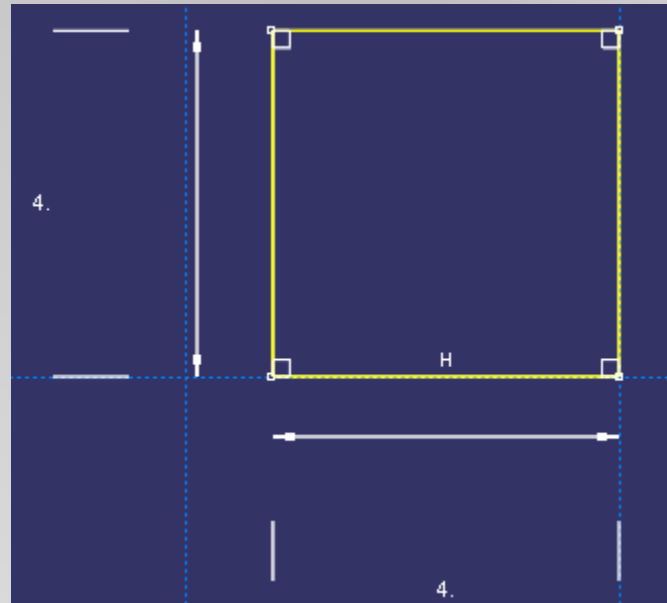
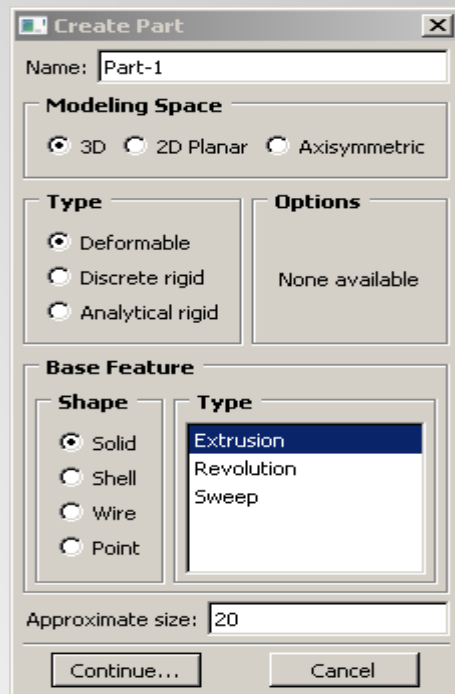
Temp = 0 degC



Temp = 100 degC



Part module



Property Module

Edit Material

Name: Aluminum

Description:

Material Behaviors

Density

Conductivity

Specific Heat

General Mechanical Thermal Other Delete

Conductivity

Type: Isotropic

☐ Use temperature-dependent data

Number of field variables: 0

Data

	Conductivity
1	210

OK Cancel

Create Section

Name: Section-1

Category

☒ Solid

☐ Shell

☐ Beam

☐ Other

Type

Homogeneous

Generalized plane strain

Continue... Cancel

Edit Section

Name: Section-1

Type: Solid, Homogeneous

Material: Aluminum Create...

Plane stress/strain thickness: 1

OK Cancel

Section Profile Composite

Manager...

Create...

Edit

Copy

Rename

Delete

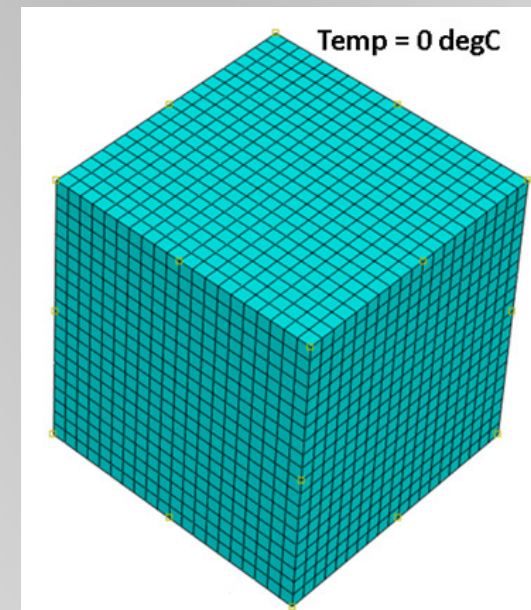
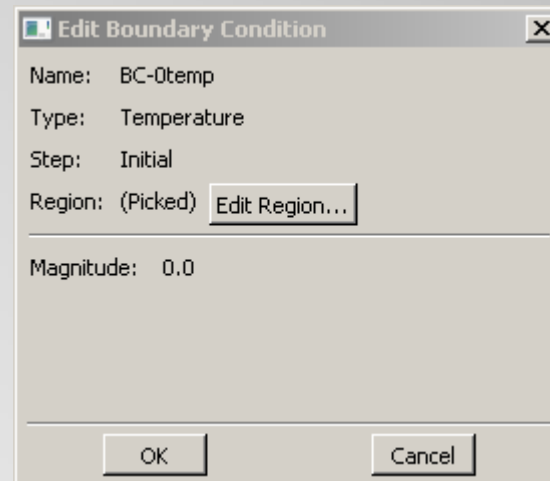
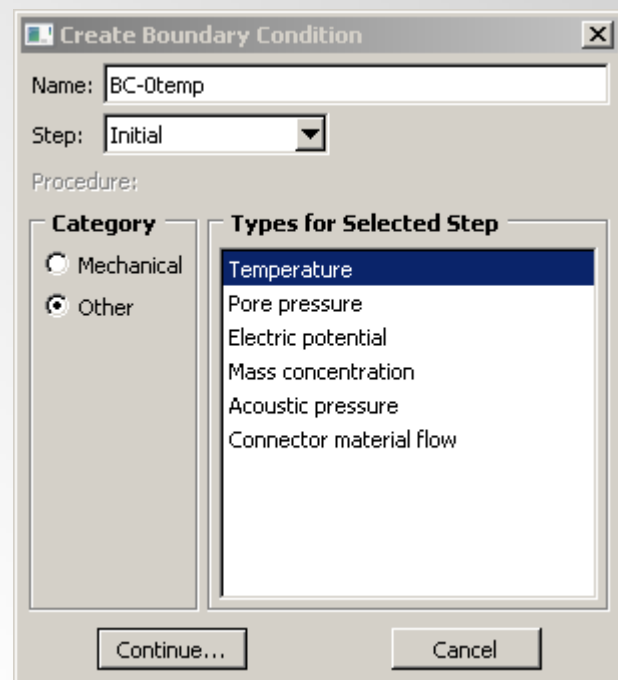
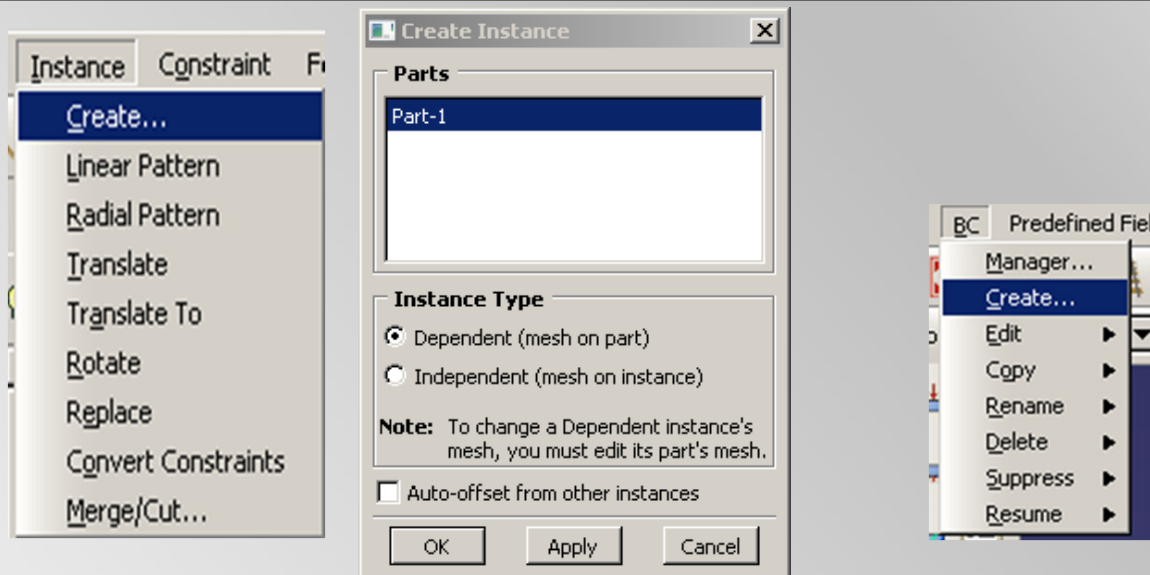
Assignment Manager...

Section Assignment Manager

Section Name (Type)	Material Name	Region
Section-1 (Solid, Homogeneous)	Aluminum	(Picked)

Create... Edit... Delete... Dismiss

Assembly & Load module: defining BC & Load



Step module

Create Step

Name:

Insert new step after

Initial

Procedure type: **General**

Dynamic, Explicit
Dynamic, Temp-disp, Explicit
Geostatic
Heat transfer
Mass diffusion
Soils
Static, General
Static, Riks

Edit Step

Name: Step-1

Type: Heat transfer

Basic Incrementation Other

Description:

Response: ☐ Steady-state ☒ Transient

Time period:

Nlgeom: Off

Edit Step

Name: Step-1

Type: Heat transfer

Basic Incrementation Other

Type: ☒ Automatic ☐ Fixed

Maximum number of increments:

Increment size:	Initial	Minimum	Maximum
	<input type="text" value="100"/>	<input type="text" value="0.01"/>	<input type="text" value="1000"/>

☒ End step when temperature change is less than:

Max. allowable temperature change per increment:

Max. allowable emissivity change per increment:

Output Other Tools Plug-ins Help

Field Output Requests ▶ Manager...
History Output Requests ▶ Create...
Integrated Output Sections ▶ **Edit** ▶ **F-Output-1**
Restart Requests... Copy
Diagnostic Print... Rename
DOF Monitor... Delete
Time Points ▶ Suppress
Resume

Edit Field Output Request

Name: F-Output-1

Step: Step-1

Procedure: Heat transfer

Domain: **Whole model**

Frequency: **Every n increments** n:

Timing: **Output at exact times**

Output Variables

☐ Select from list below ☒ Preselected defaults ☐ All ☐ Edit variables

HFL,NT,RFL,

☐ Displacement/Velocity/Acceleration
☐ Energy
☒ Thermal
☐ Porous media/Fluids

Note: Error indicators are not available when Domain is Whole Model or Interaction.

Load module: defining BC & Load

Create Boundary Condition

Name: BC-100temp

Step: Step-1

Procedure: Heat transfer

Category

☐ Mechanical

☒ Other

Types for Selected Step

Temperature

Connector material flow

Submodel

Continue... Cancel

Edit Boundary Condition

Name: BC-100temp

Type: Temperature

Step: Step-1 (Heat transfer)

Region: (Picked) Edit Region...

Distribution: Uniform Create...

Magnitude: 100

Amplitude: (Instantaneous) Create...

OK Cancel

Make sure BC-0temp is inactive (not propagated) in Step-1

Boundary Condition Manager

	Name	Initial	Step-1
✓	BC-0temp	Created	Inactive
✓	BC-100temp		Created

Edit...

Move Left

Move Right

Activate

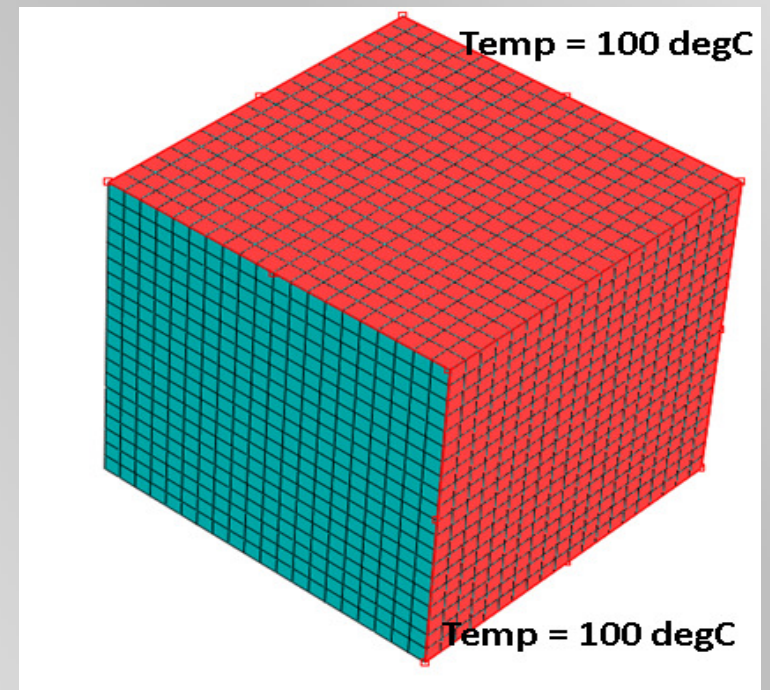
Deactivate

Step procedure: Heat transfer

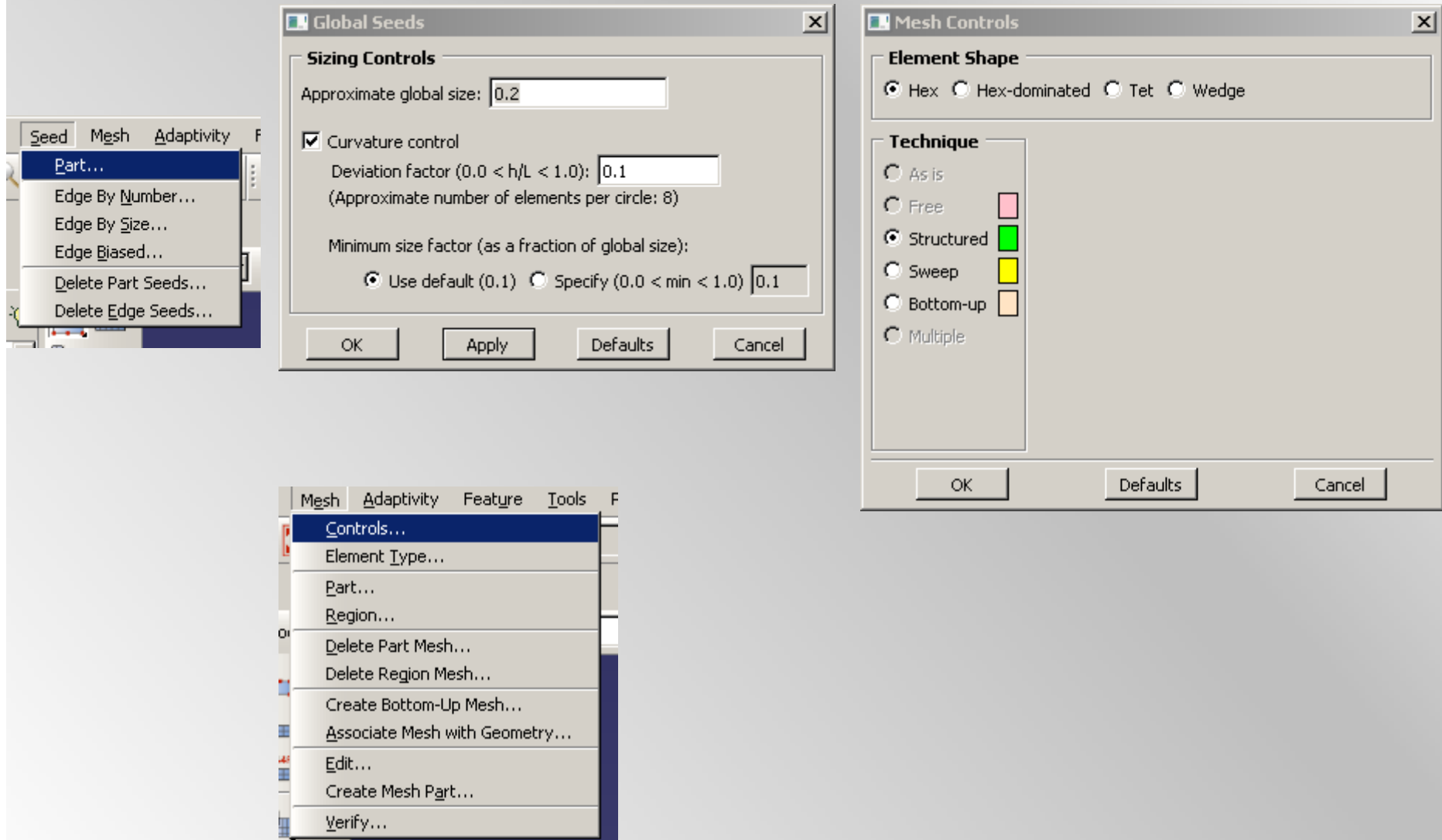
Boundary condition type: Temperature

Boundary condition status: Created in this step

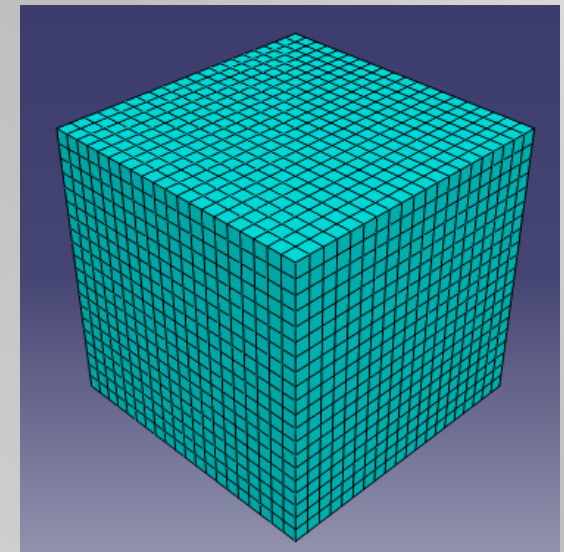
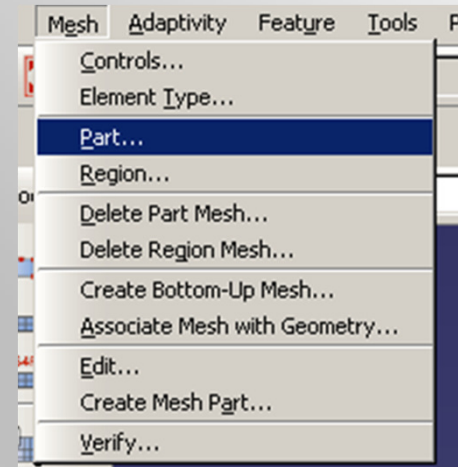
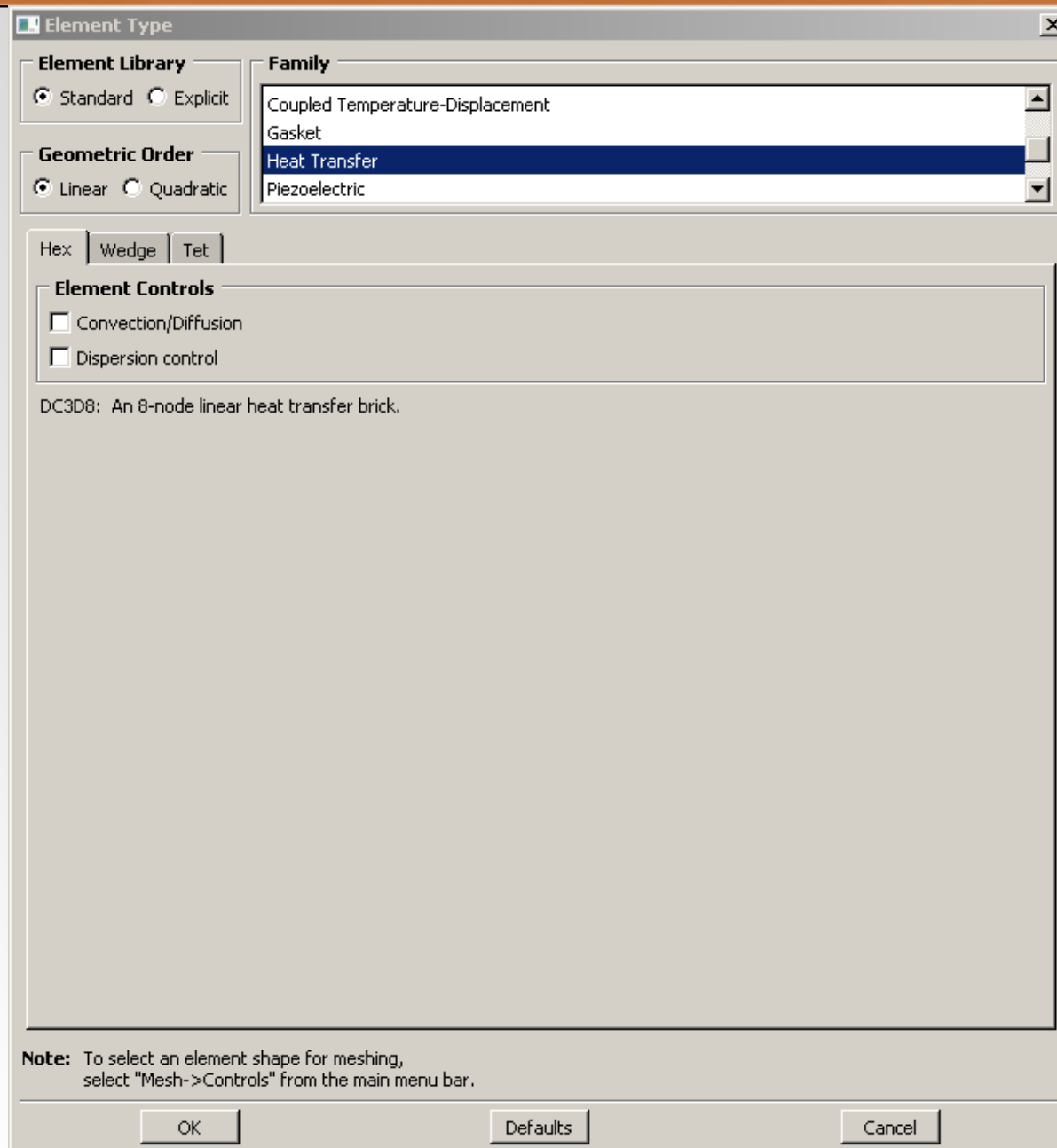
Create... Copy... Rename... Delete... Dismiss



Mesh Module



Mesh Module



Job Module

Job Manager

Name	Model	Type	Status
------	-------	------	--------

Buttons: Write Input, Data Check, Submit, Continue, Monitor..., Results, Kill

Buttons: Create..., Edit..., Copy..., Rename..., Delete..., Dismiss

Create Job

Name:

Source:

Model-1

Buttons: Continue..., Cancel

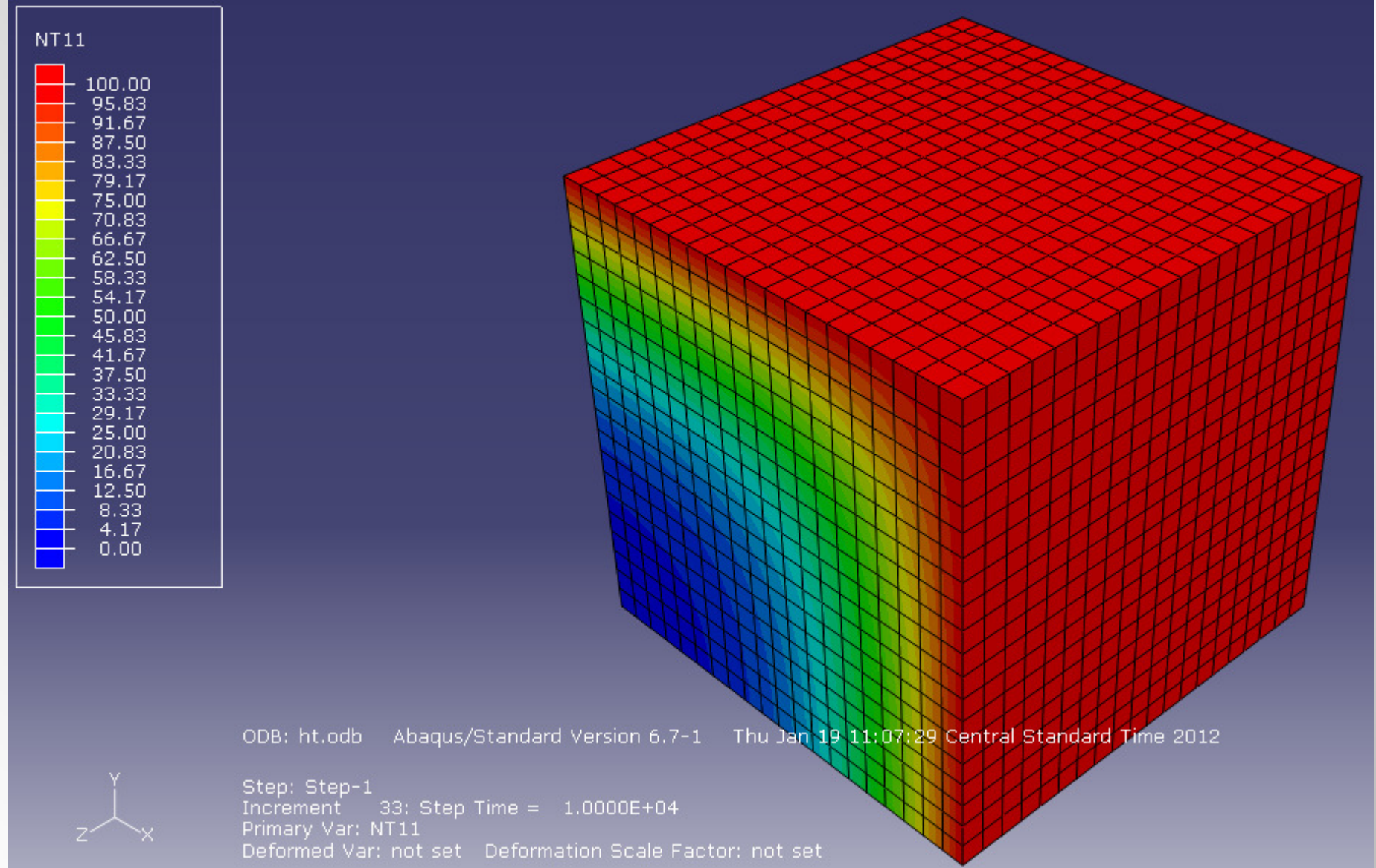
Job Manager

Name	Model	Type	Status
ht	Model-1	Full Analysis	None

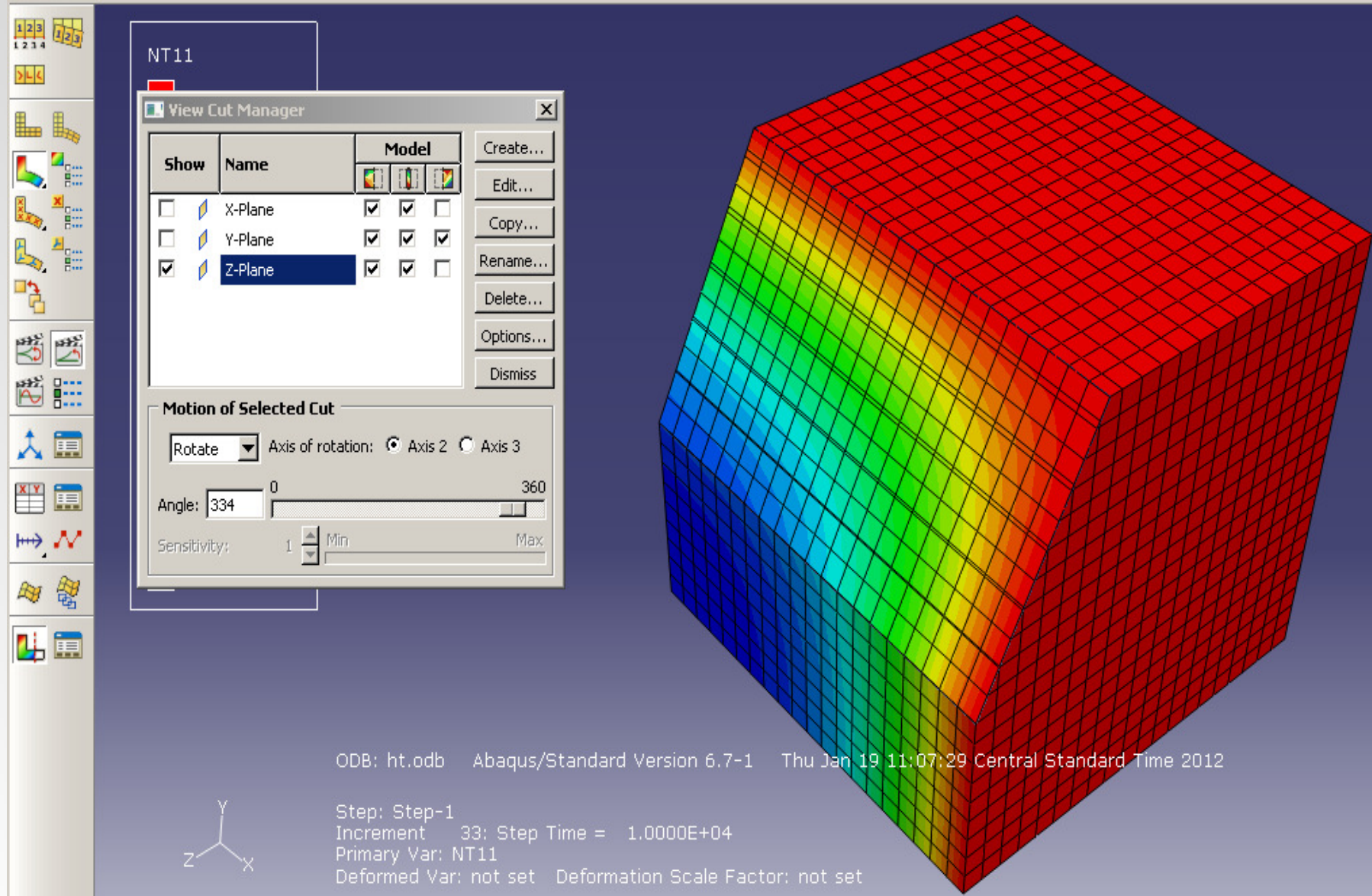
Buttons: Write Input, Data Check, Submit, Continue, Monitor..., Results, Kill

Buttons: Create..., Edit..., Copy..., Rename..., Delete..., Dismiss

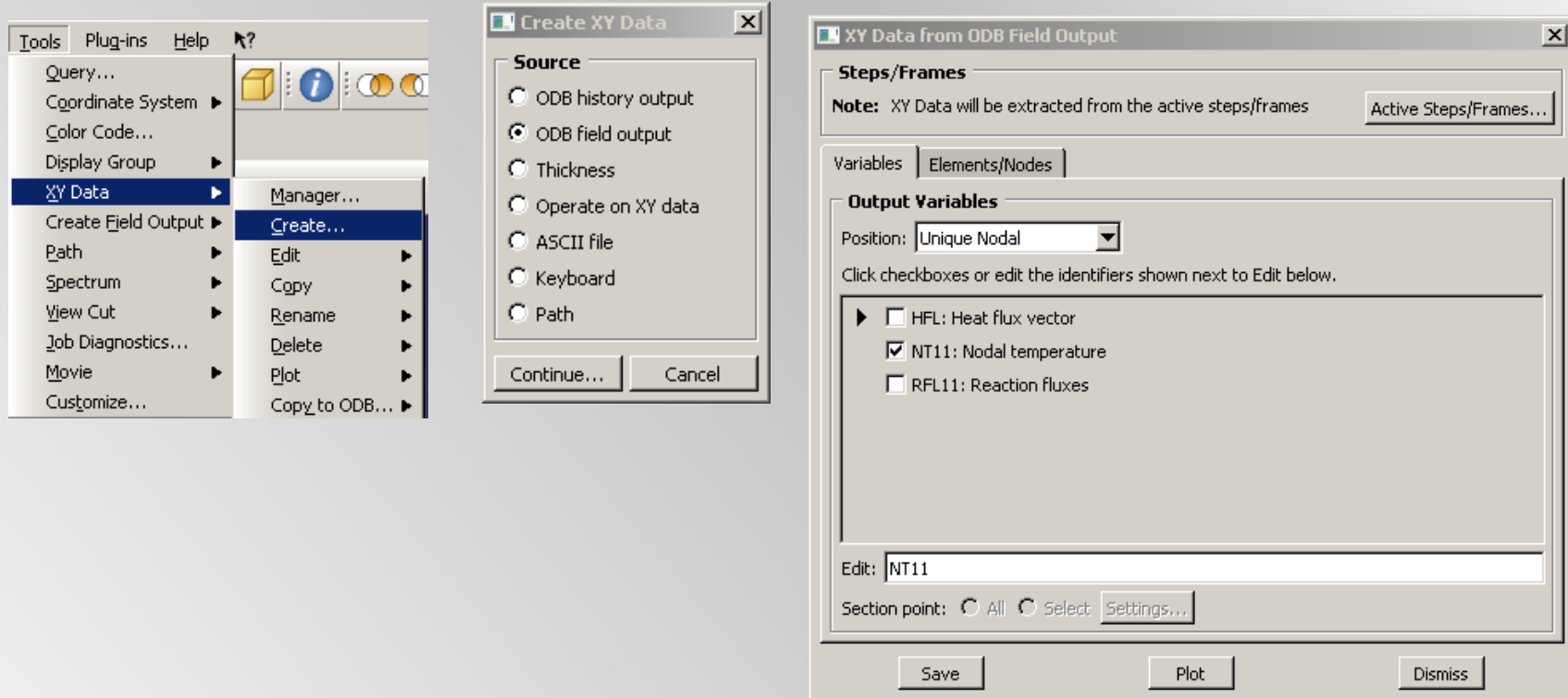
Visualization Module : Temperature



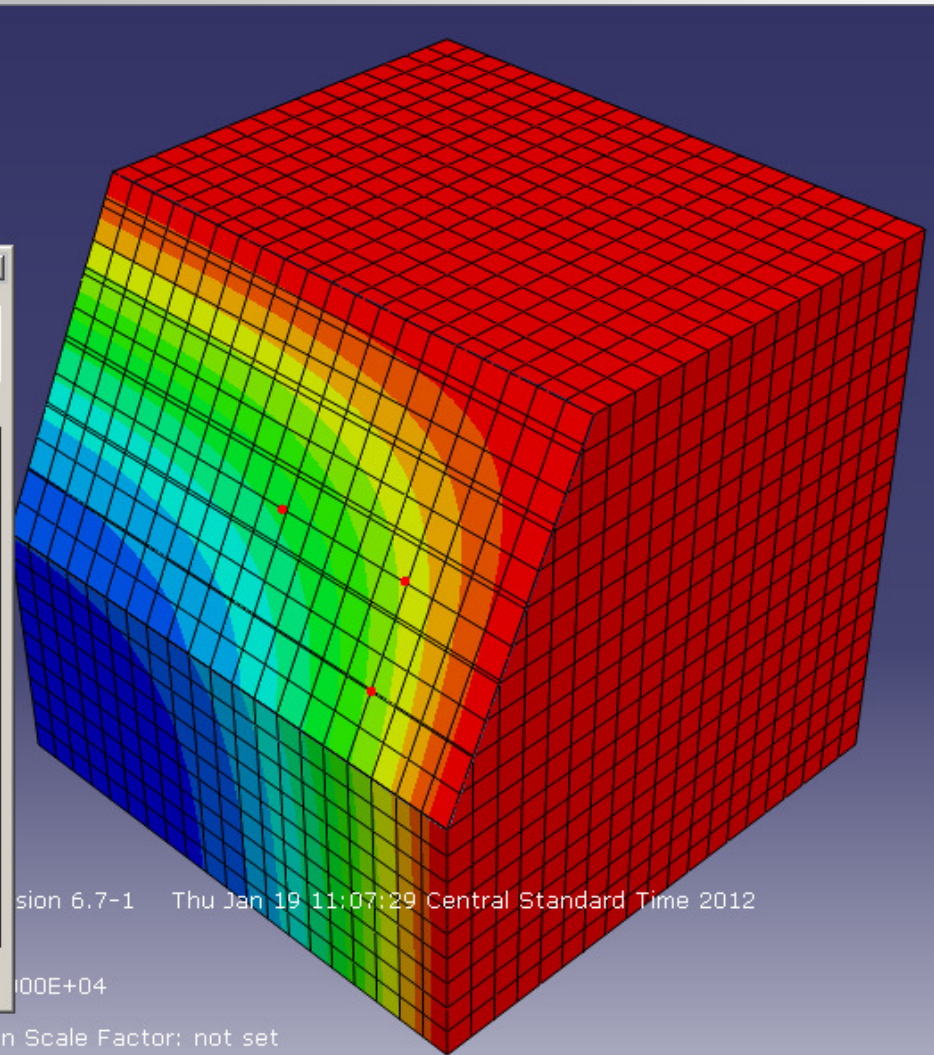
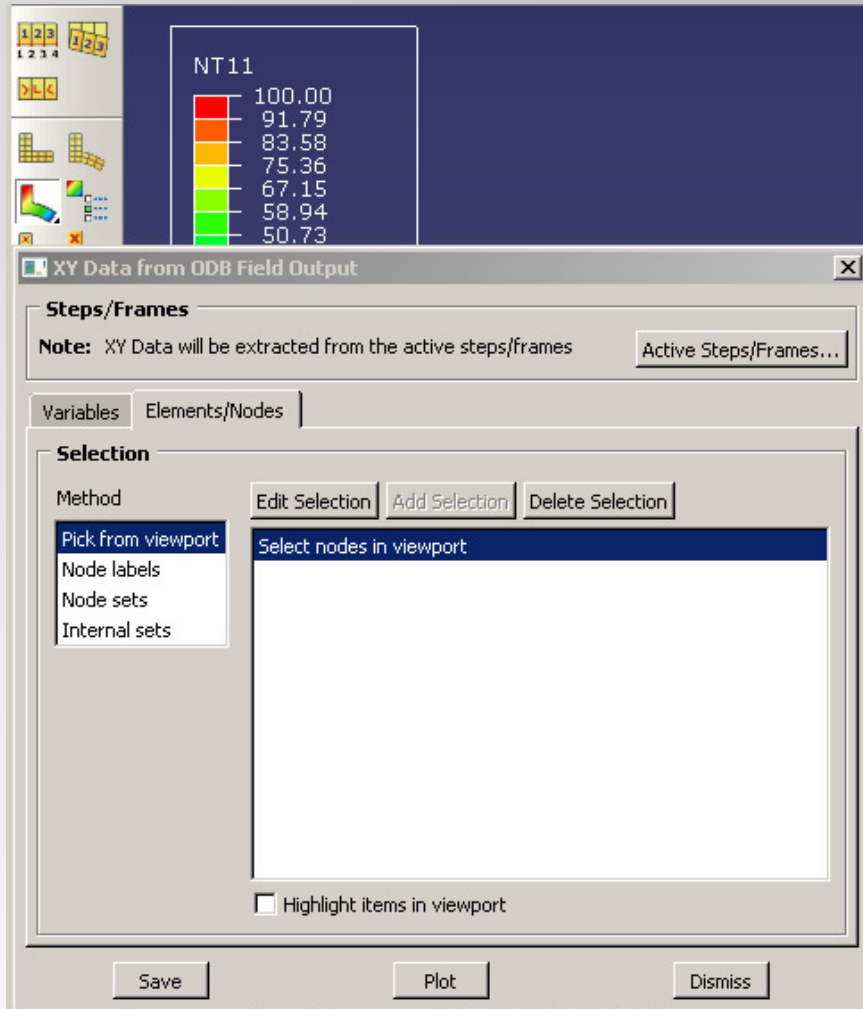
Visualization Module : Section cut



Visualization Module : Create XY data



Visualization Module : Create XY data

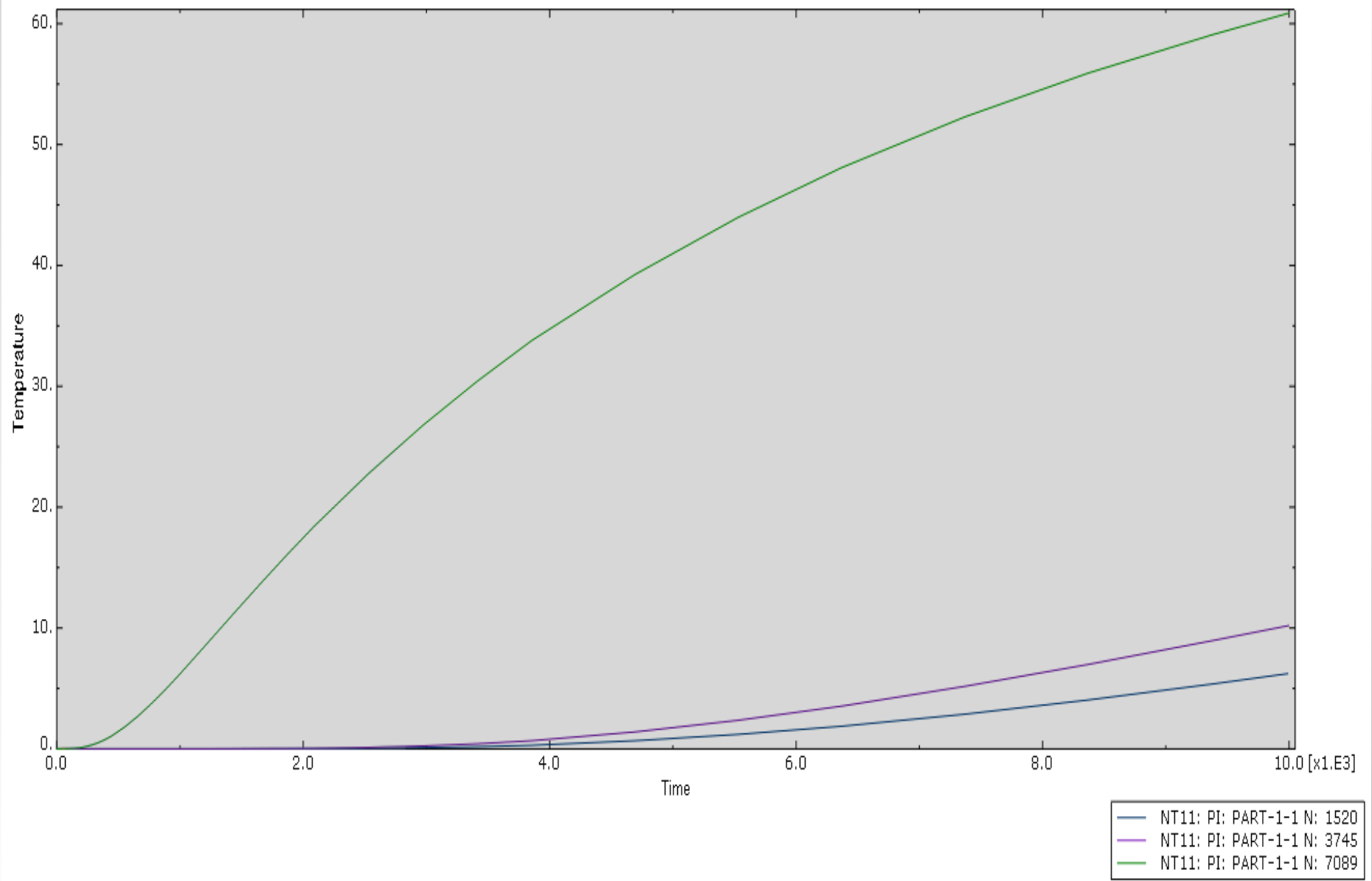



session 6.7-1 Thu Jan 19 11:07:29 Central Standard Time 2012

00E+04

Deformed Var: not set Deformation Scale Factor: not set

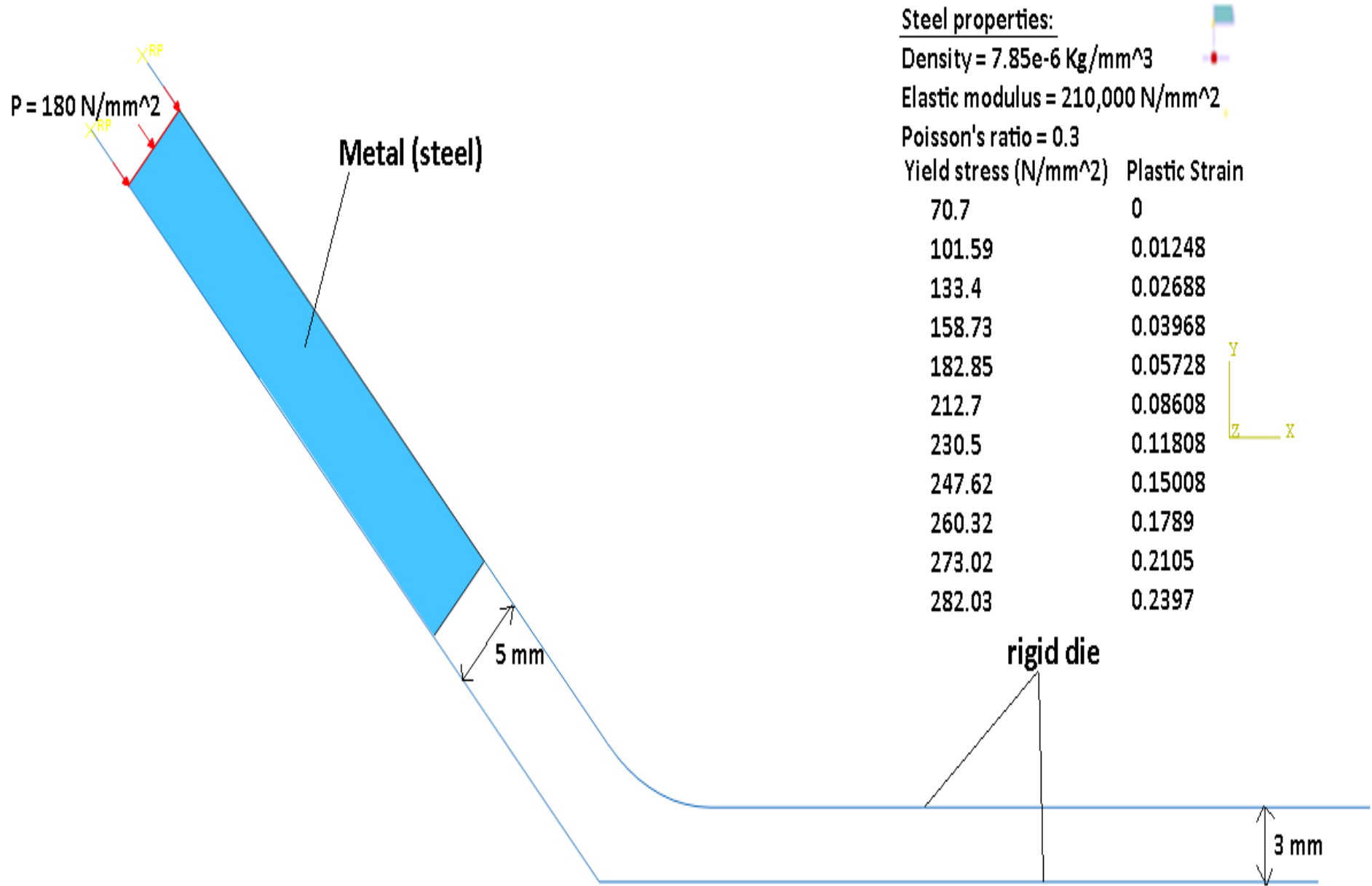
Visualization Module : XY Plot



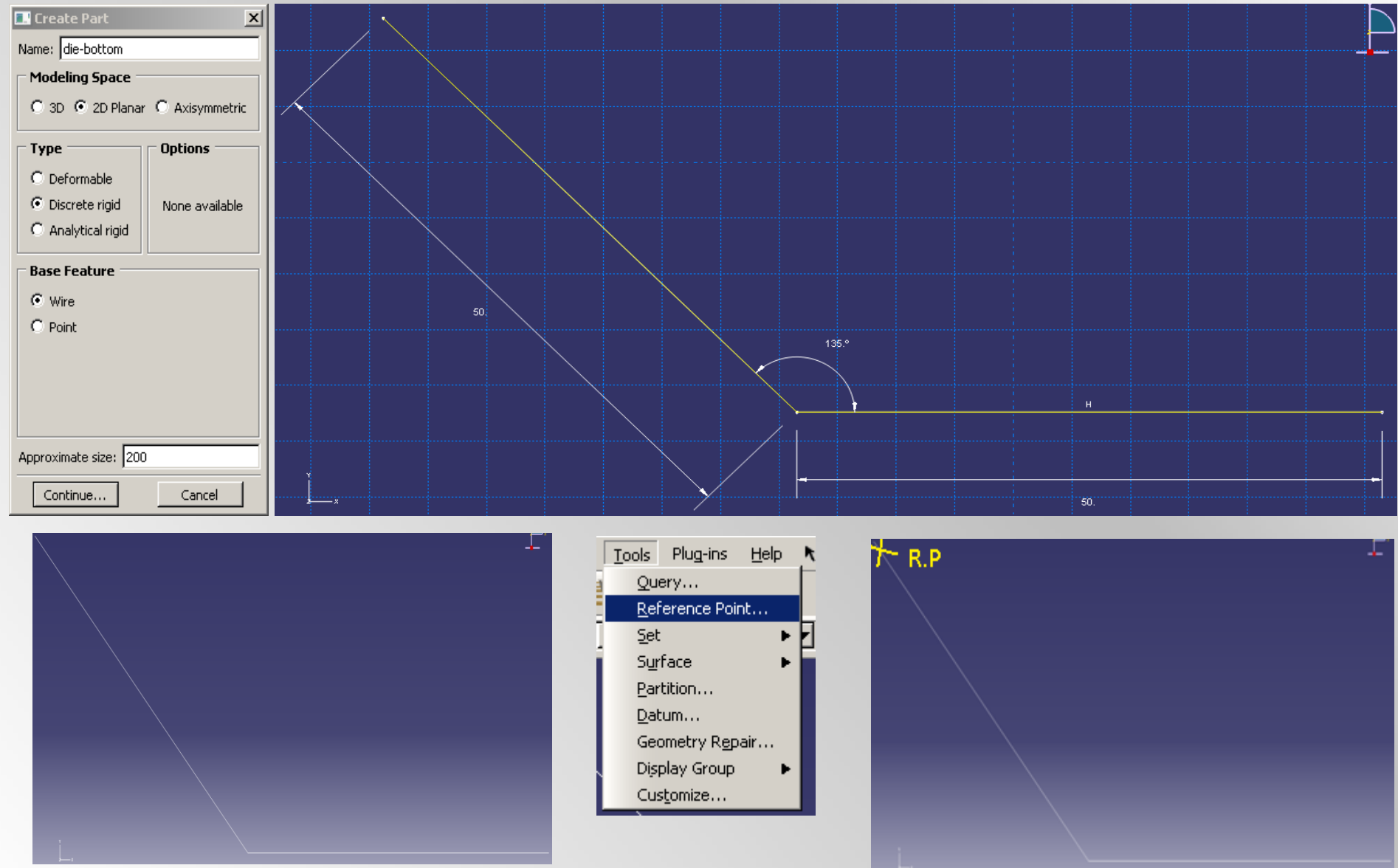


Example 3: Extrusion of metal (Dynamic explicit analysis)

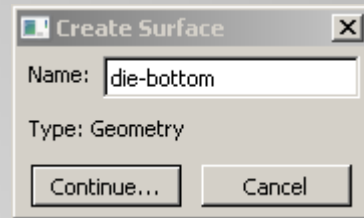
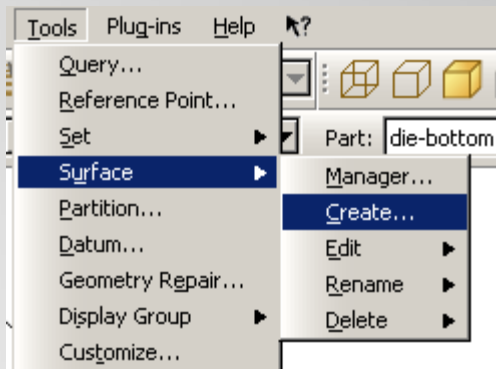
Problem description



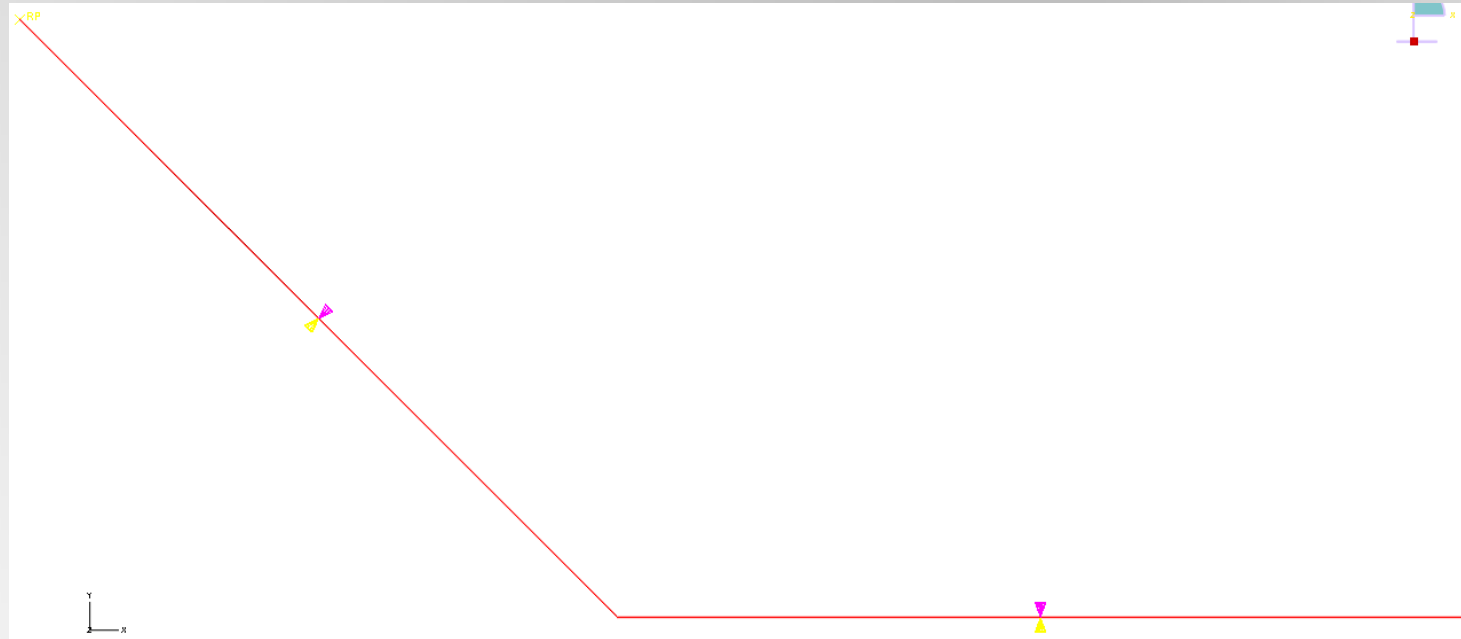
Part module



Part module: Surface creation

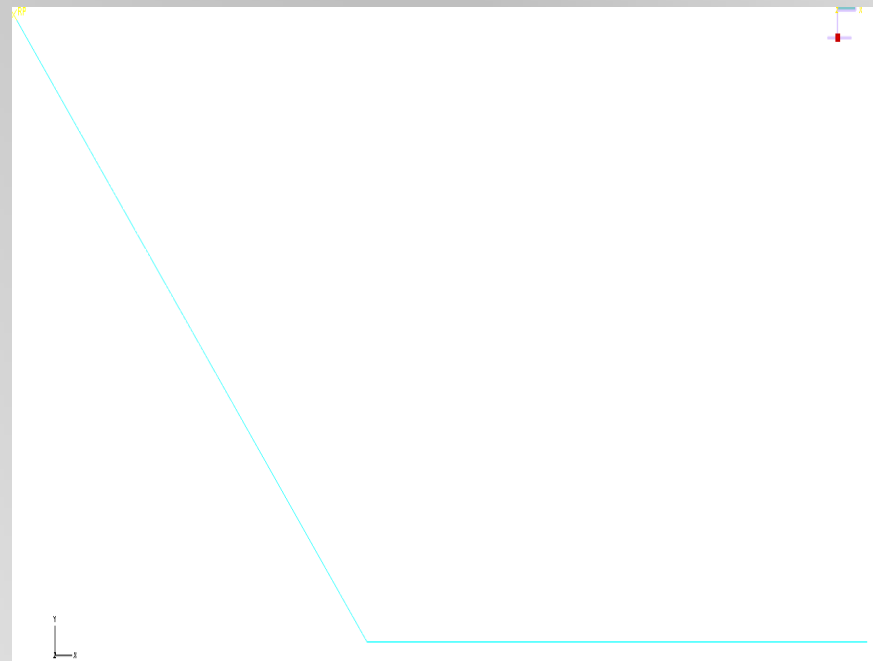
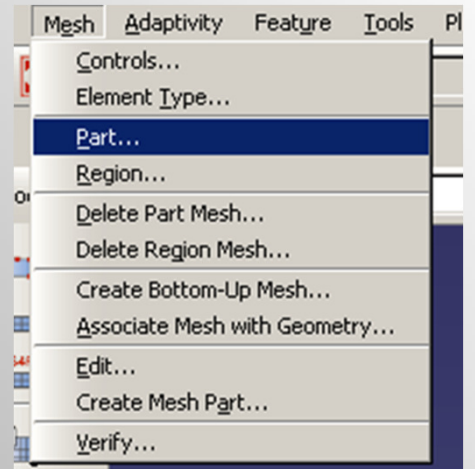
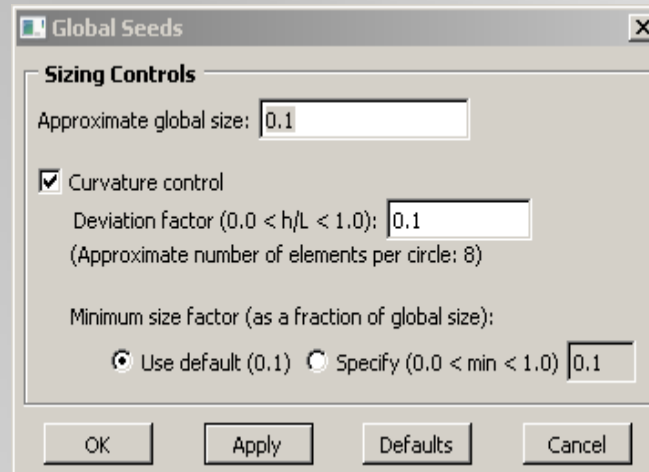
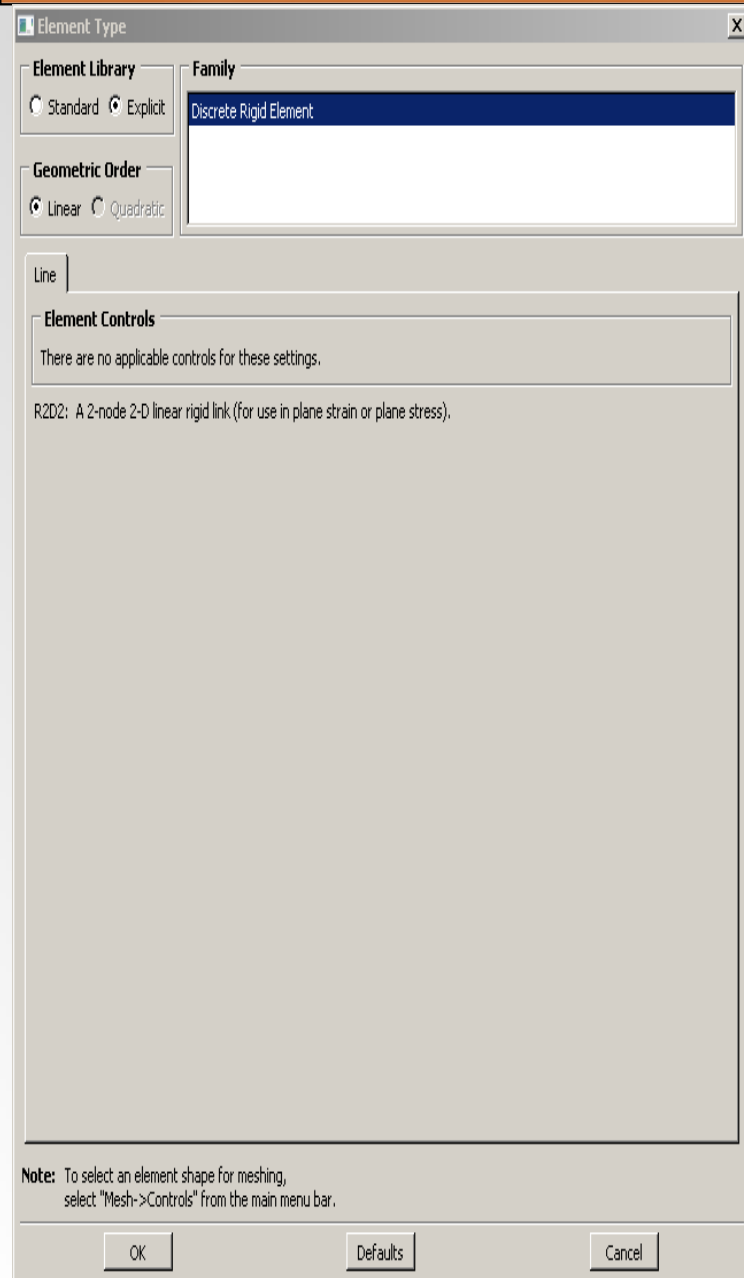


This surface will be later used in “Interaction module” to define contact between different surfaces

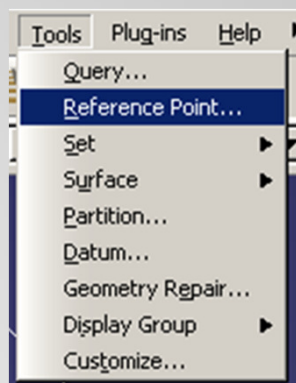
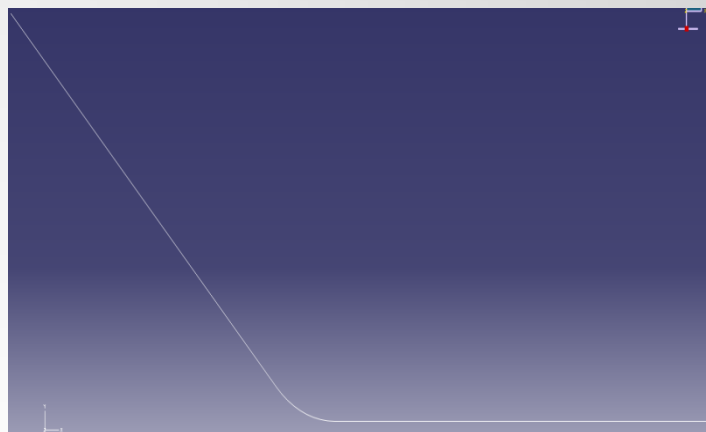
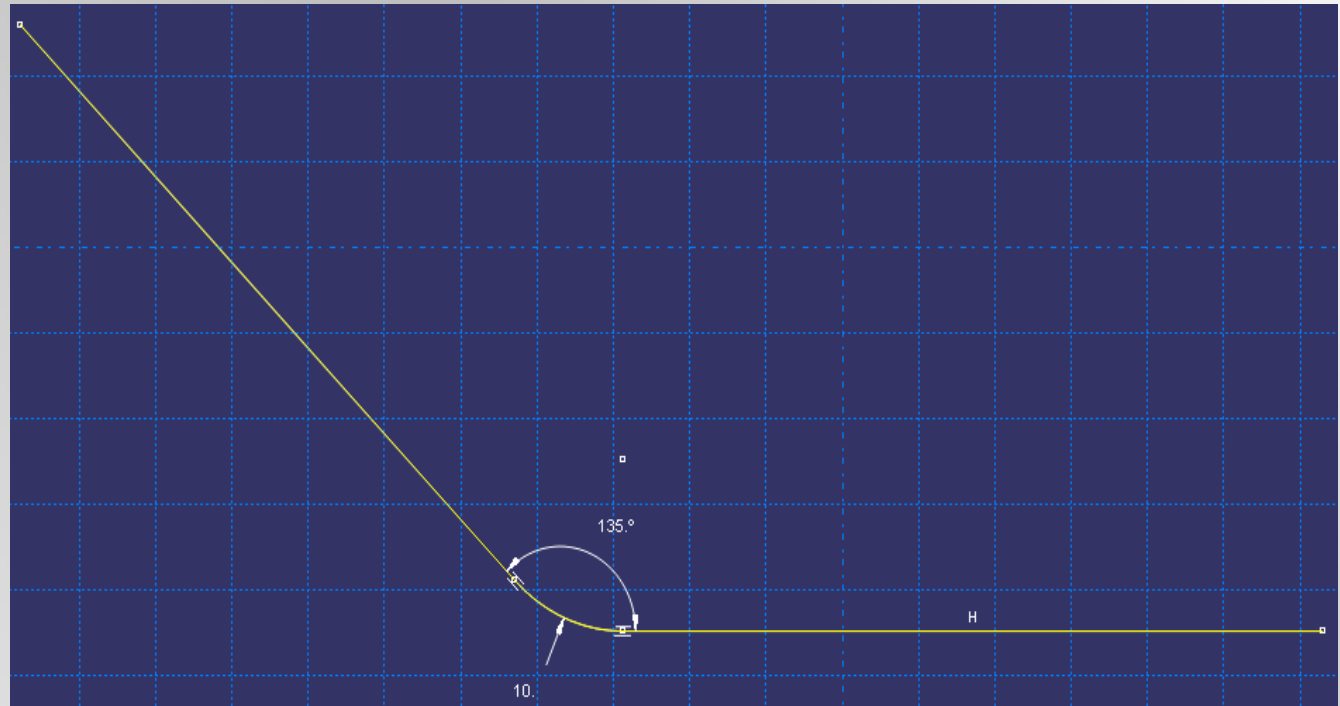
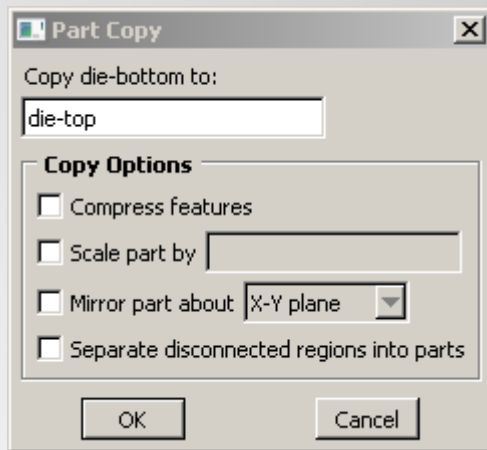
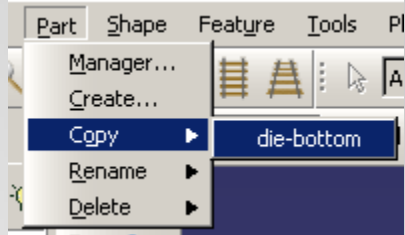


Choose magenta as metal is going to come in contact with top surface of the bottom die

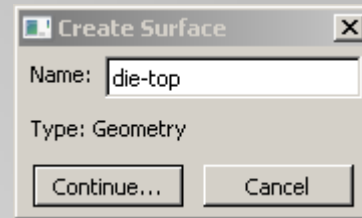
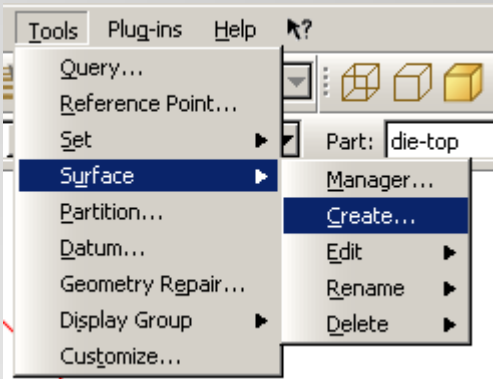
Mesh module



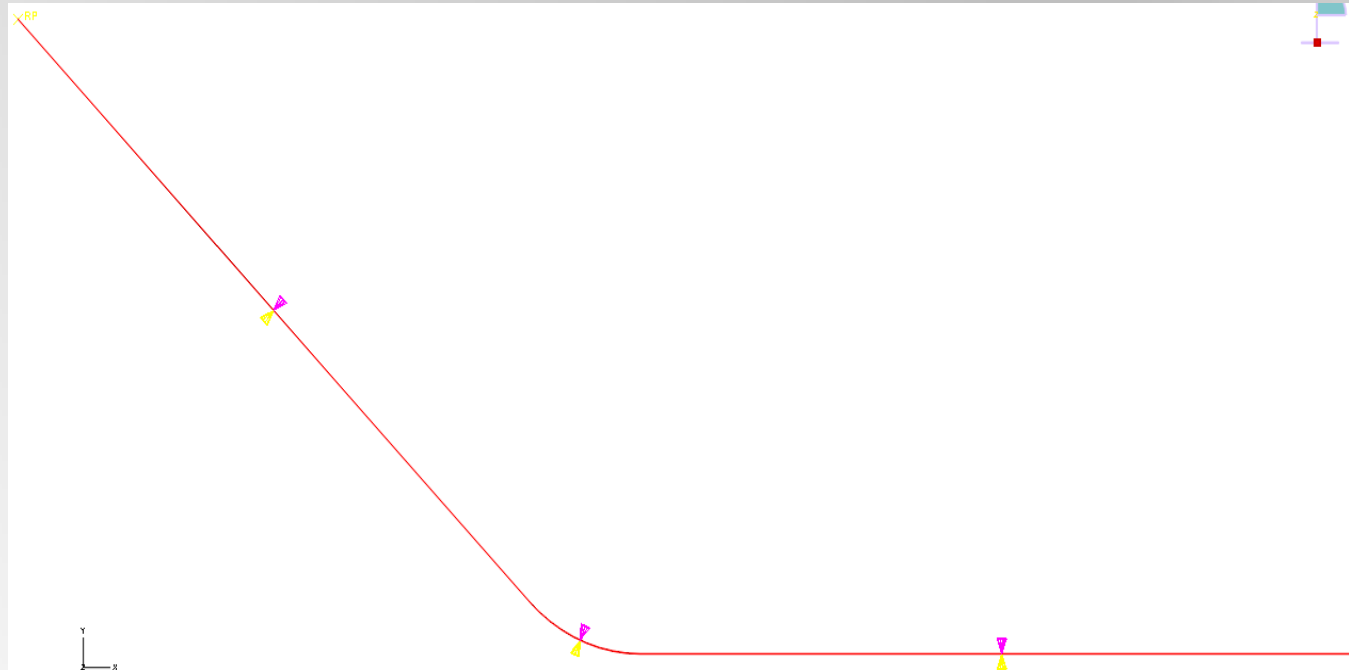
Part module: create new part by copying old part



Part module: Surface Creation

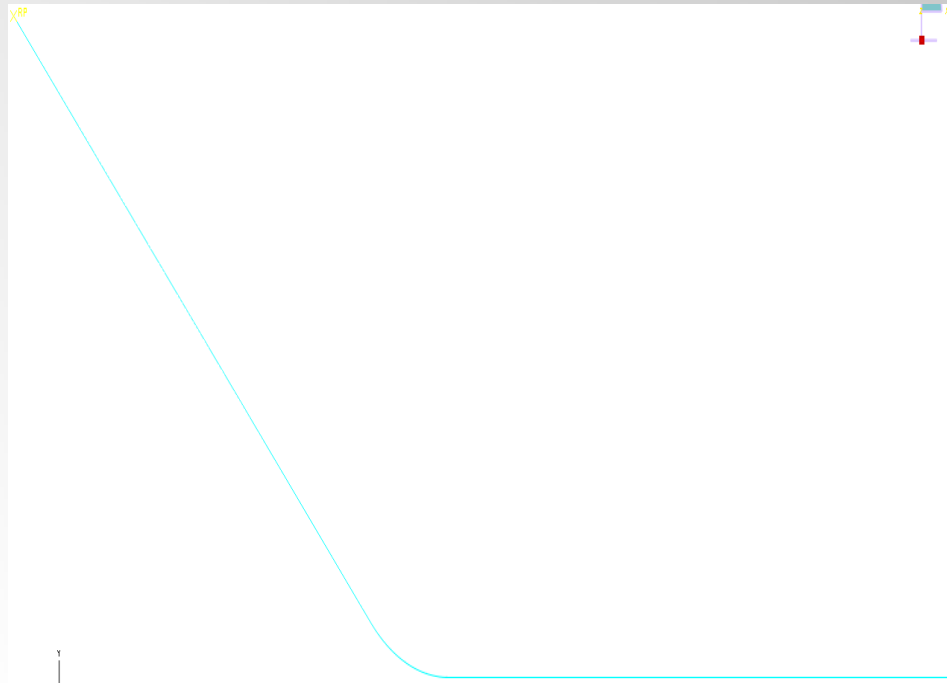
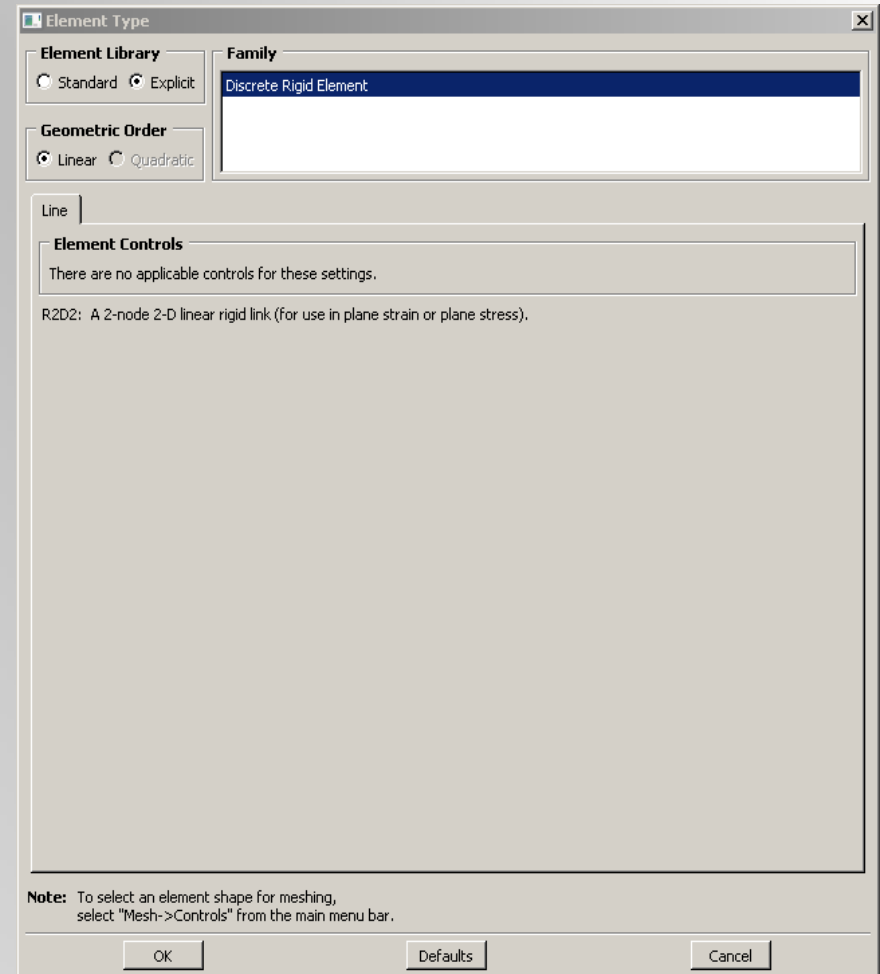
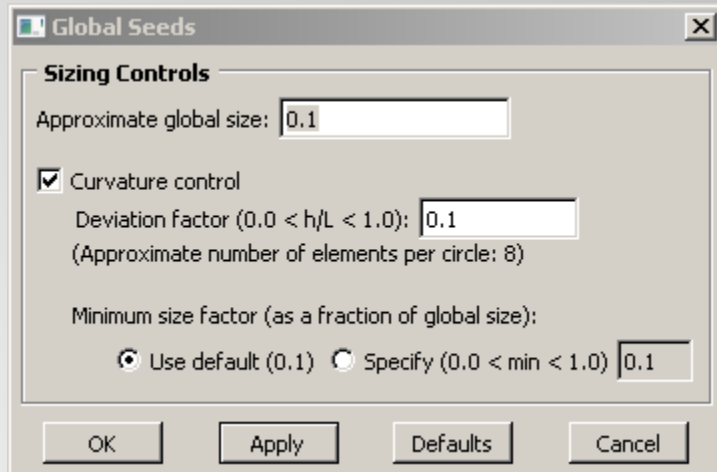


This surface will be later used in “Interaction module” to define contact between different surfaces



Choose yellow as metal is going to come in contact with bottom surface of the top die

Mesh module



Part module

Create Part

Name: metal

Modeling Space

☐ 3D ☒ 2D Planar ☐ Axisymmetric

Type

☒ Deformable
☐ Discrete rigid
☐ Analytical rigid

Options

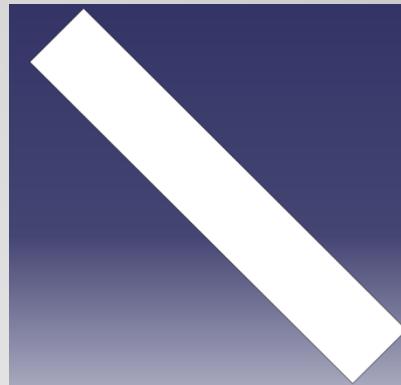
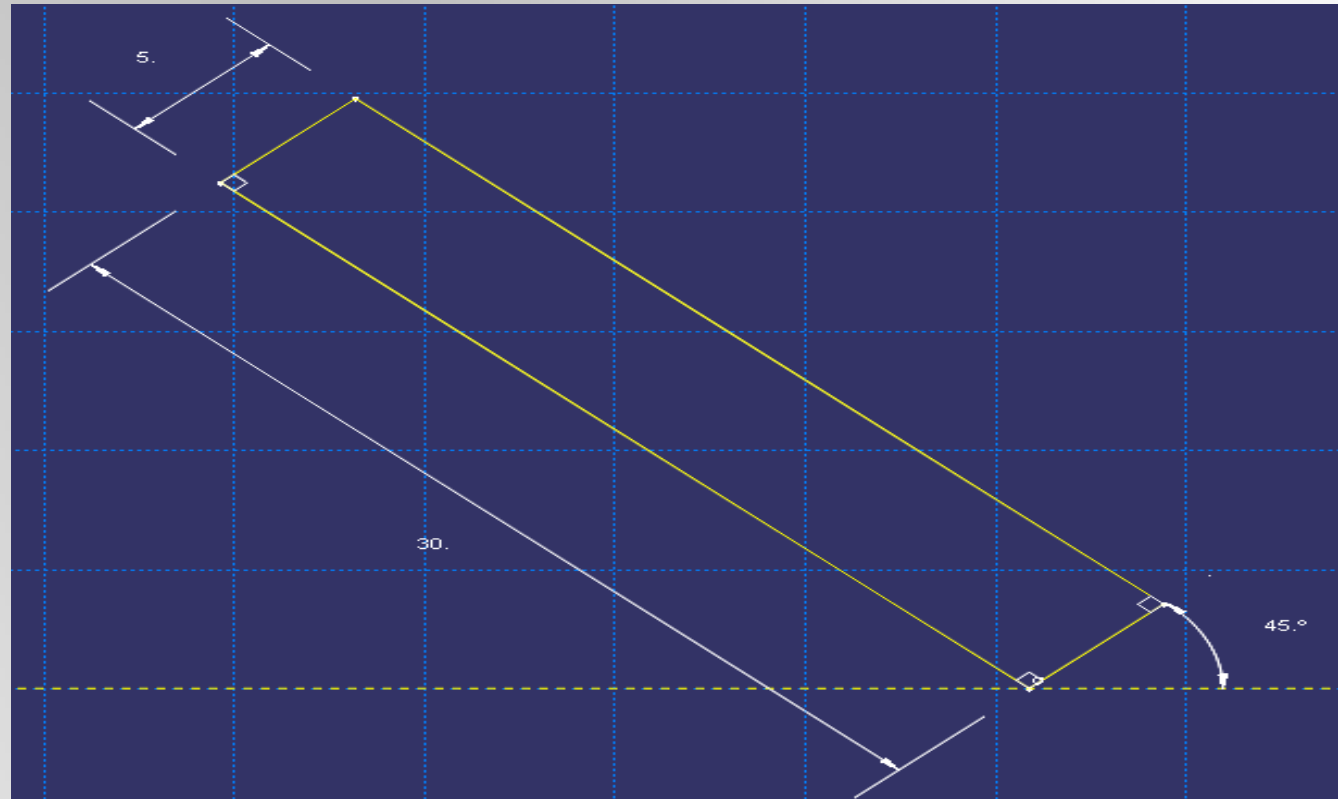
None available

Base Feature

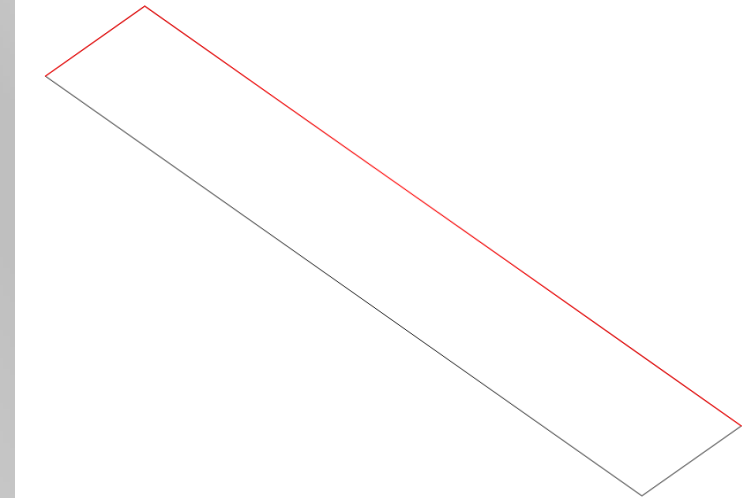
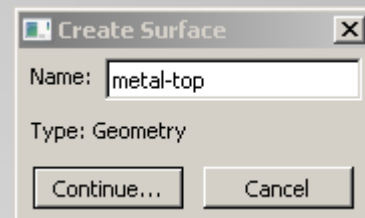
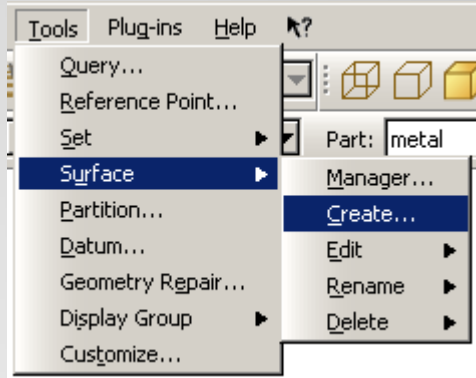
☒ Shell
☐ Wire
☐ Point

Approximate size: 200

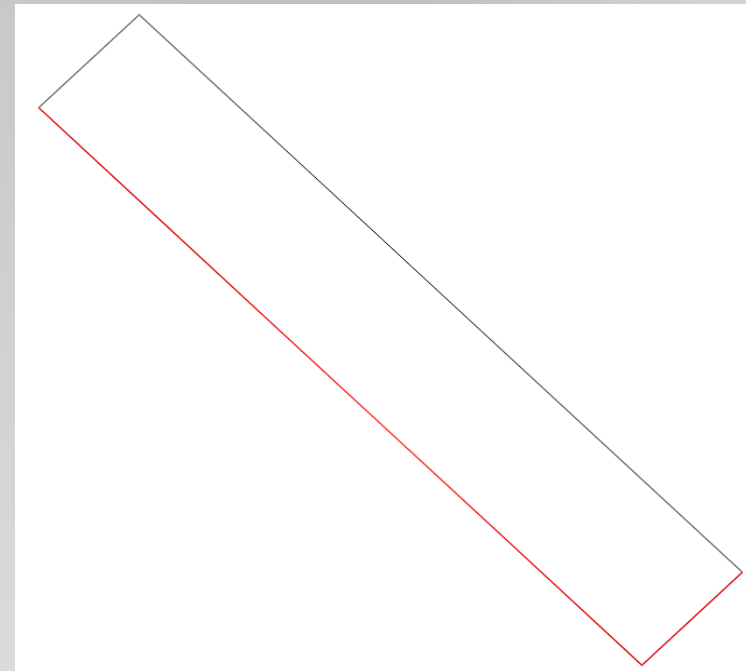
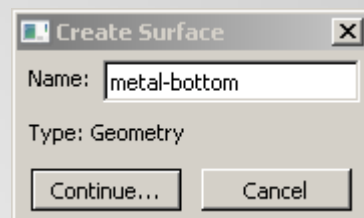
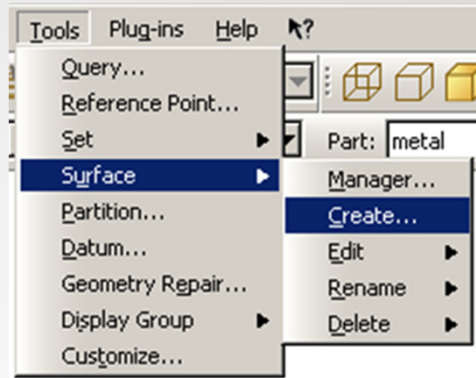
Continue... Cancel



Part module: Surface Creation



These surfaces will be later used in "Interaction module" to define contact between different surfaces



Mesh module

Global Seeds

Sizing Controls

Approximate global size:

☒ Curvature control
 Deviation factor ($0.0 < h/L < 1.0$):
 (Approximate number of elements per circle: 8)

Minimum size factor (as a fraction of global size):
☒ Use default (0.1) ☐ Specify ($0.0 < \text{min} < 1.0$)

OK Apply Defaults Cancel

Mesh Controls

Element Shape
☒ Quad ☐ Quad-dominated ☐ Tri

Technique
☐ As is
☐ Free
☒ Structured
☐ Sweep
☐ Multiple

Algorithm Options
☒ Minimize the mesh transition [Tip...](#)

Redefine Region Corners...

OK Defaults Cancel

Element Type

Element Library
☐ Standard ☒ Explicit

Geometric Order
☒ Linear ☐ Quadratic

Family
 Cohesive
 Coupled Temperature-Displacement
 Plane Strain
 Plane Stress

Quad Tri

Element Controls

Second-order accuracy: ☐ Yes ☒ No

Distortion control: ☒ Use default ☐ Yes ☐ No
 Length ratio:

Hourglass control: ☒ Use default ☐ Enhanced ☐ Relax stiffness ☐ Stiffness ☐ Viscous ☐ Combined
 Stiffness-viscous weight factor:

Element deletion: ☒ Use default ☐ Yes ☐ No

Max Degradation: ☒ Use default ☐ Specify

Displacement hourglass scaling factor:

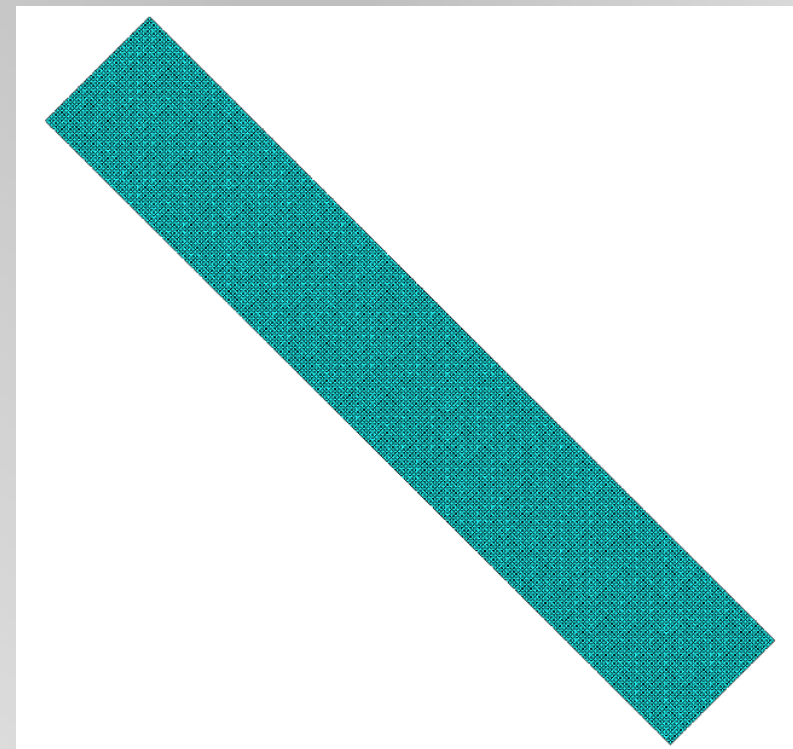
Linear bulk viscosity scaling factor:

Quadratic bulk viscosity scaling factor:

CP54R: A 4-node bilinear plane stress quadrilateral, reduced integration, hourglass control.

Note: To select an element shape for meshing, select "Mesh->Controls" from the main menu bar.

OK Defaults Cancel



Property module

Edit Material

Name: Steel

Description:

Material Behaviors

Density

Elastic

Plastic

General Mechanical Thermal Other

Plastic

Hardening: Isotropic

☐ Use strain-rate-dependent data

☐ Use temperature-dependent data

Number of field variables: 0

Data

	Yield Stress	Plastic Strain
1	70.7	0
2	101.59	0.01248
3	133.4	0.02688
4	158.73	0.03968
5	182.85	0.05728
6	212.7	0.08608
7	230.5	0.11808
8	247.62	0.15008

OK Cancel

Create Section

Name: Section-1

Category

☒ Solid

☐ Shell

☐ Beam

☐ Other

Type

Homogeneous

Generalized plane strain

Continue... Cancel

Section Profile Composite

Manager...

Create...

Edit

Copy

Rename

Delete

Assignment Manager...

Section Assignment Manager

Section Name (Type)	Material Name	Region
Section-1 (Solid, Homogeneous)	Steel	(Picked)

Create... Edit... Delete... Dismiss

Assembly module

Create Instance

Parts

- die-bottom
- die-top
- metal

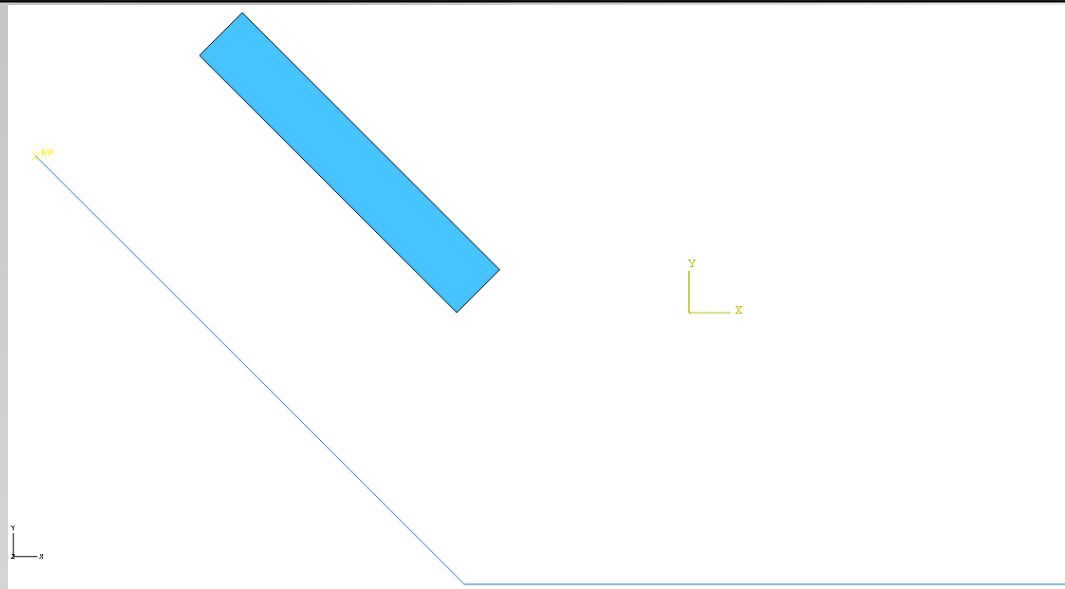
Instance Type

A meshed part has been selected, so the instance type will be Dependent.

Note: To change a Dependent instance's mesh, you must edit its part's mesh.

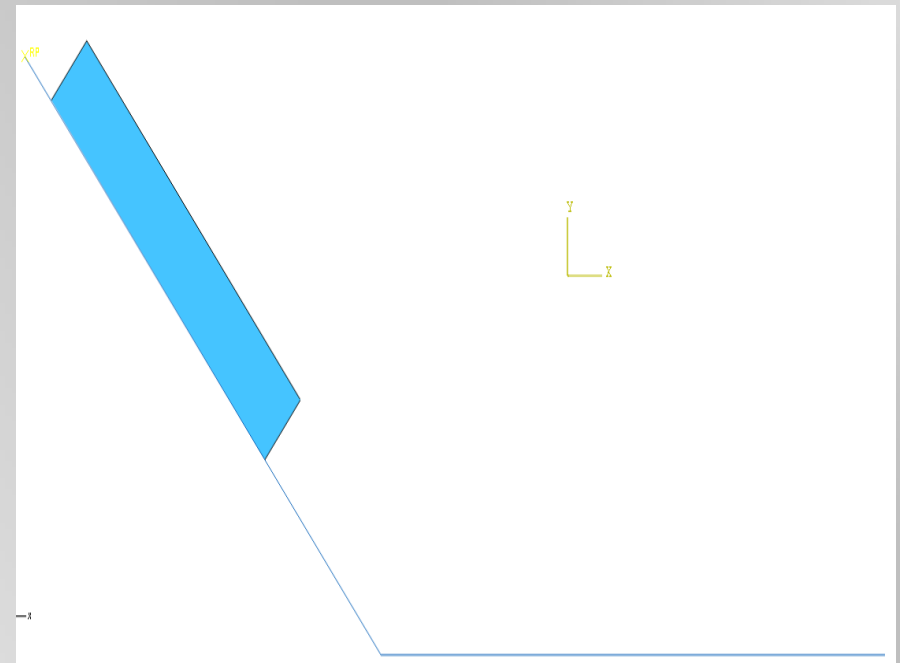
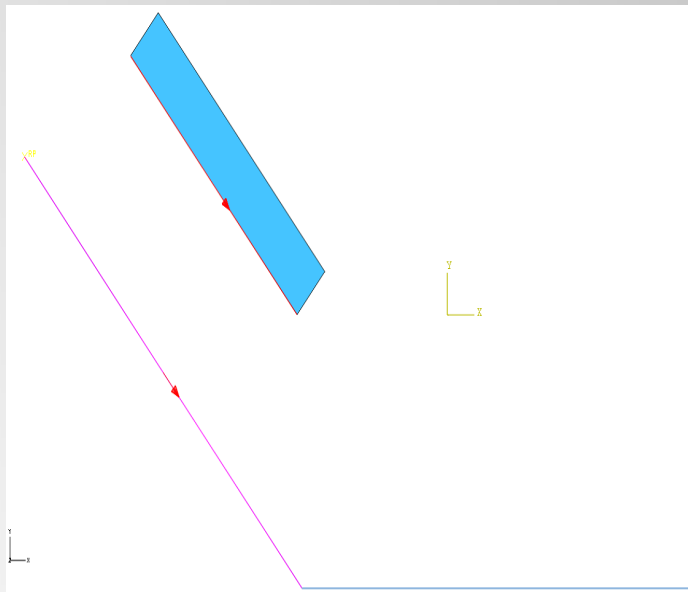
☐ Auto-offset from other instances

OK Apply Cancel



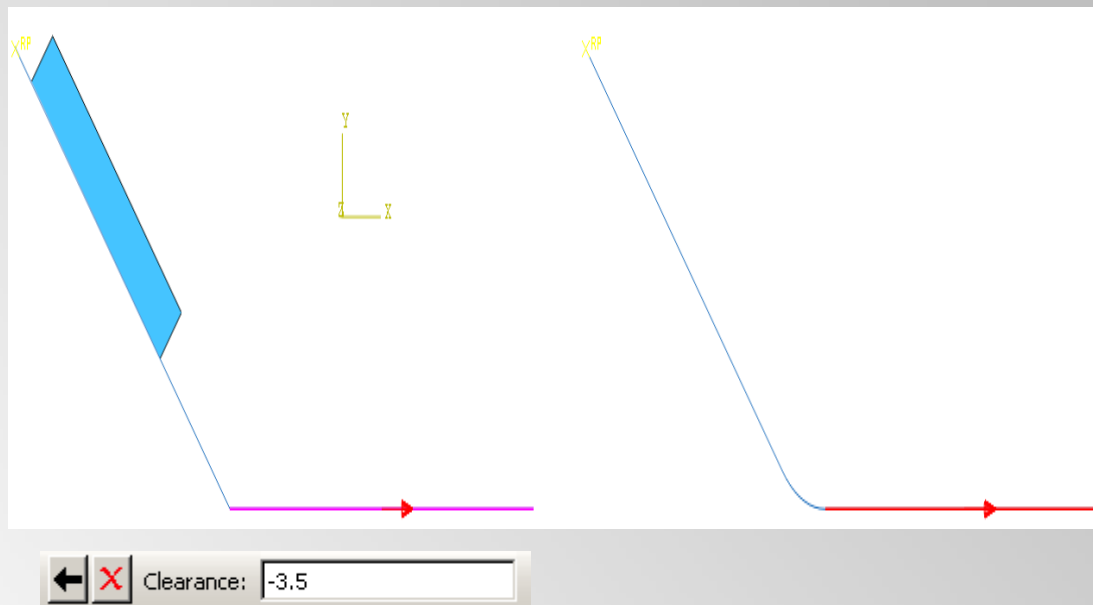
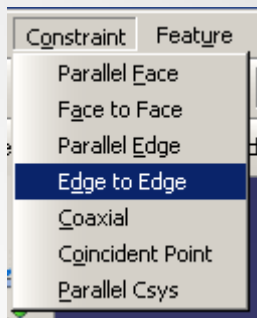
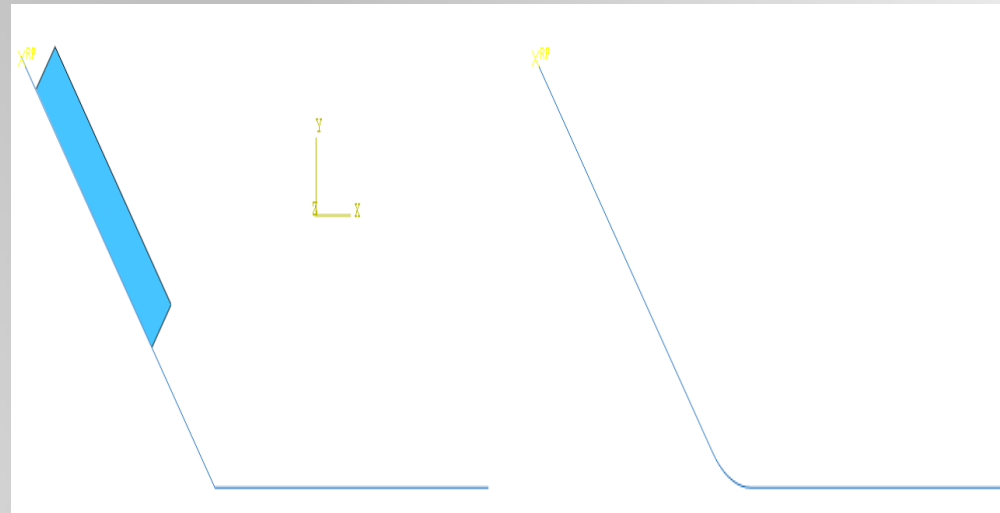
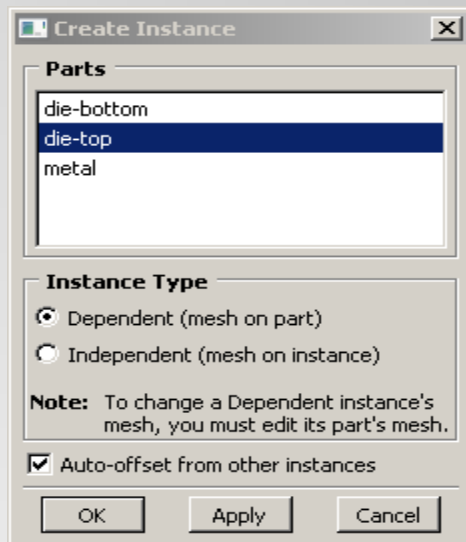
Constraint Feature

- Parallel Face
- Face to Face
- Parallel Edge
- Edge to Edge
- Coaxial
- Coincident Point
- Parallel Csys

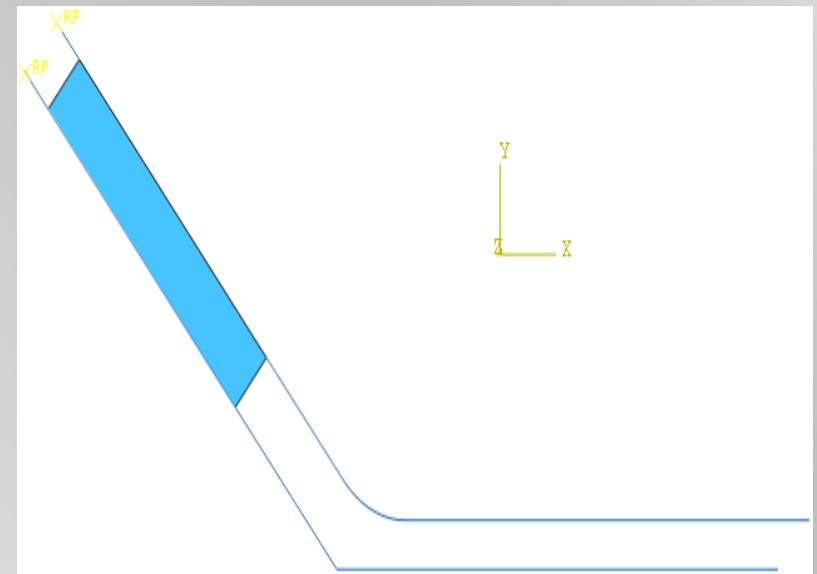
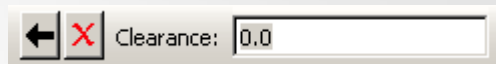
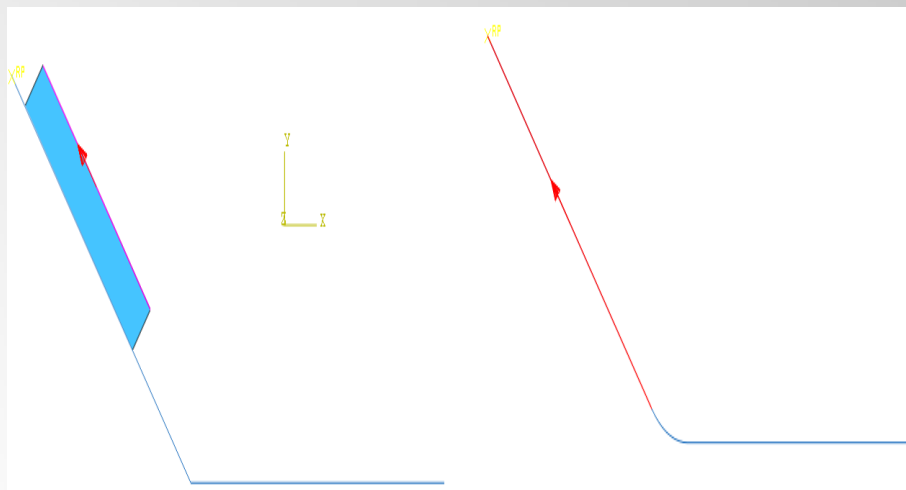
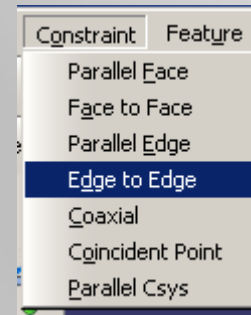
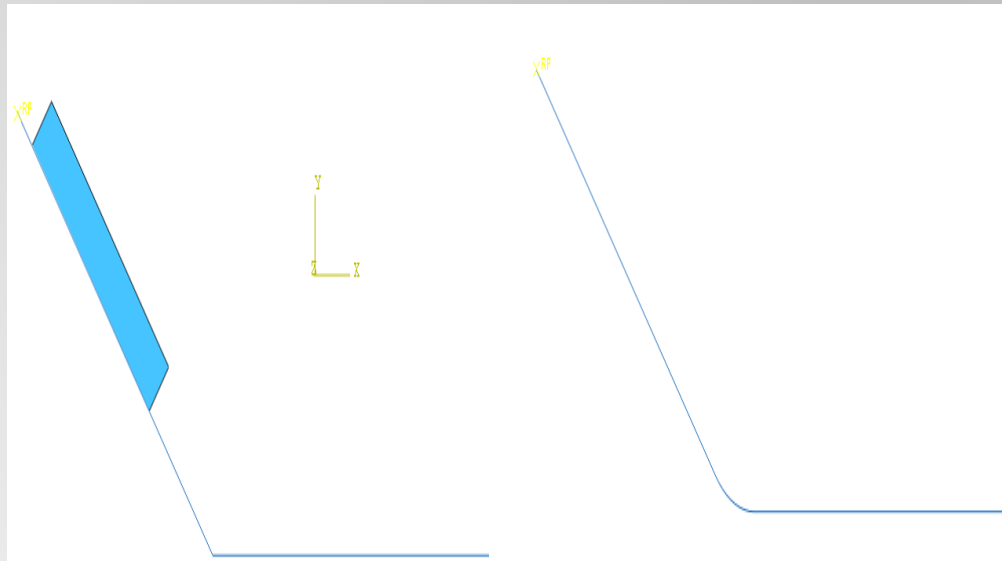


← X Clearance: 0.0

Assembly module



Assembly module



Step module

Create Step

Name: Step-1

Insert new step after

Initial

Procedure type: General

Dynamic, Explicit

Dynamic, Temp-disp, Explicit

Geostatic

Heat transfer

Mass diffusion

Soils

Static, General

Static, Riks

Continue...

Cancel

Edit Step

Name: Step-1

Type: Dynamic, Explicit

Basic Incrementation Mass scaling Other

Description:

Time period: 0.015

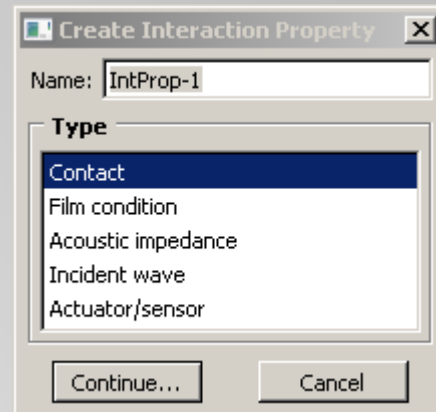
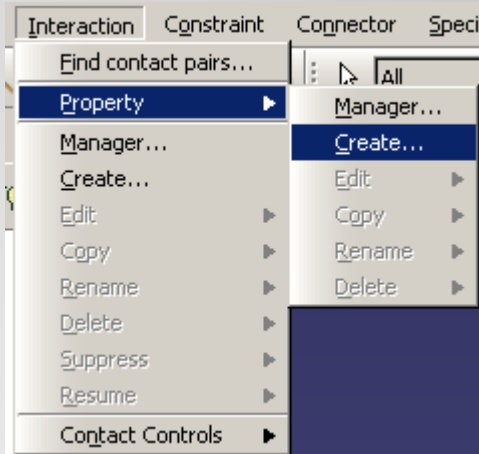
Nlgeom: On

☐ Include adiabatic heating effects

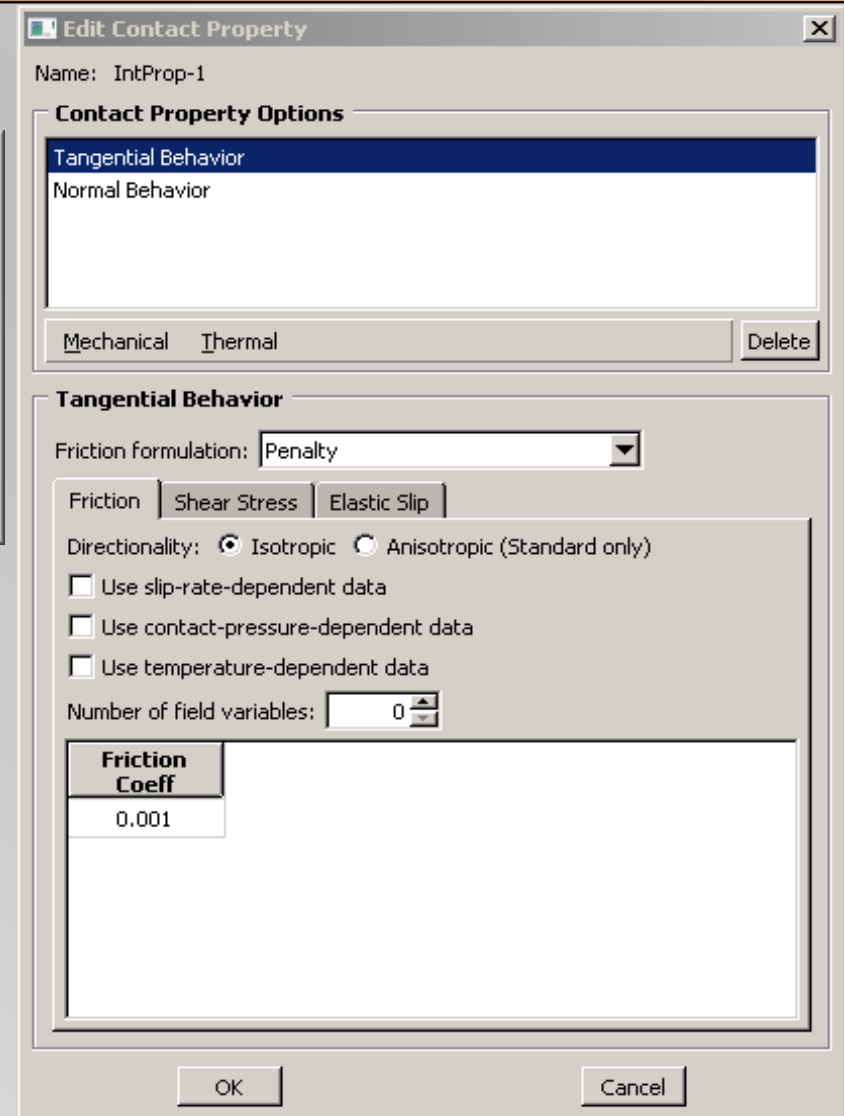
OK

Cancel

Interaction module



Refer section
15.14.1 of CAE
user's manual



Interaction module

The screenshot displays the Interaction module interface. On the left, a menu is open with 'Create...' selected. Below it, the 'Create Interaction' dialog box is shown with 'Int-1' as the name and 'Initial' as the step. The 'Types for Selected Step' list includes 'Surface-to-surface contact (Explicit)', which is highlighted. The main area shows a 2D model of a mechanical part with a blue rectangular region and a red line. The 'Edit Interaction' dialog box is open on the right, showing details for 'Int-1'. It specifies 'Surface-to-surface contact (Explicit)' type, 'Initial' step, and 'Penalty contact method' for the mechanical constraint formulation. The 'Sliding formulation' is set to 'Finite sliding'. The 'Clearance' section includes a note: 'Note: Clearance can only be used with small sliding in the first analysis step.' The 'Contact interaction property' is 'IntProp-1' and the 'Weighting factor' is 'Use analysis default'. The 'Contact controls' are set to '(Default)'. The 'OK' and 'Cancel' buttons are at the bottom of the dialog.

Interaction Constraint

Find contact pairs...
Property
Manager...
Create...
Edit
Copy
Rename
Delete
Suppress
Resume
Contact Controls

Create Interaction

Name: Int-1
Step: Initial
Procedure:
Types for Selected Step
General contact (Explicit)
Surface-to-surface contact (Explicit)
Self-contact (Explicit)
Acoustic impedance
Continue... Cancel

Edit Interaction

Name: Int-1
Type: Surface-to-surface contact (Explicit)
Step: Initial
First surface: die-bottom-1.die-bottom Edit Region... Switch
Second surface: metal-1.bottom Edit Region...
Mechanical constraint formulation: Penalty contact method
Sliding formulation: ☒ Finite sliding ☐ Small sliding
Clearance
Note: Clearance can only be used with small sliding in the first analysis step.
Contact interaction property: IntProp-1 Create...
Weighting factor: ☒ Use analysis default ☐ Specify
Contact controls: (Default)
OK Cancel

Refer section
15.13.6 of CAE
user's manual

Interaction module

The screenshot displays the Interaction module interface. On the left, a context menu is open with the 'Create...' option selected. Below it, the 'Create Interaction' dialog box is shown, with 'Name: Int-2' and 'Step: Initial' set. The 'Types for Selected Step' list includes 'Surface-to-surface contact (Explicit)', which is highlighted. On the right, the 'Edit Interaction' dialog box for 'Int-2' is open. It shows 'Type: Surface-to-surface contact (Explicit)', 'Step: Initial', and 'First surface: die-top-1.die-top'. The 'Second surface' is 'metal-1.metal-top'. The 'Mechanical constraint formulation' is set to 'Penalty contact method'. The 'Sliding formulation' has 'Finite sliding' selected. The 'Clearance' section contains a note: 'Note: Clearance can only be used with small sliding in the first analysis step.' The 'Contact interaction property' is 'IntProp-1', and the 'Weighting factor' is 'Use analysis default'. The 'Contact controls' are set to '(Default)'. The background shows a 2D model of a mechanical part with a blue rectangular region and a red curved line, with various points marked by small squares. A coordinate system (X, Y, Z) is visible at the bottom left.

Interaction Constraint

Find contact pairs...
Property
Manager...
Create...
Edit
Copy
Rename
Delete
Suppress
Resume
Contact Controls

Create Interaction

Name: Int-2
Step: Initial
Procedure:
Types for Selected Step
General contact (Explicit)
Surface-to-surface contact (Explicit)
Self-contact (Explicit)
Acoustic impedance
Continue... Cancel

Edit Interaction

Name: Int-2
Type: Surface-to-surface contact (Explicit)
Step: Initial
First surface: die-top-1.die-top Edit Region...
Second surface: metal-1.metal-top Edit Region... Switch
Mechanical constraint formulation: Penalty contact method
Sliding formulation: ☒ Finite sliding ☐ Small sliding
Clearance
Note: Clearance can only be used with small sliding in the first analysis step.
Contact interaction property: IntProp-1 Create...
Weighting factor: ☒ Use analysis default ☐ Specify
Contact controls: (Default)
OK Cancel

Load module: Boundary conditions

Create Boundary Condition

Name: BC-1

Step: Initial

Procedure:

Category

☒ Mechanical

☐ Other

Types for Selected Step

Symmetry/Antisymmetry/Encastre

Displacement/Rotation

Velocity/Angular velocity

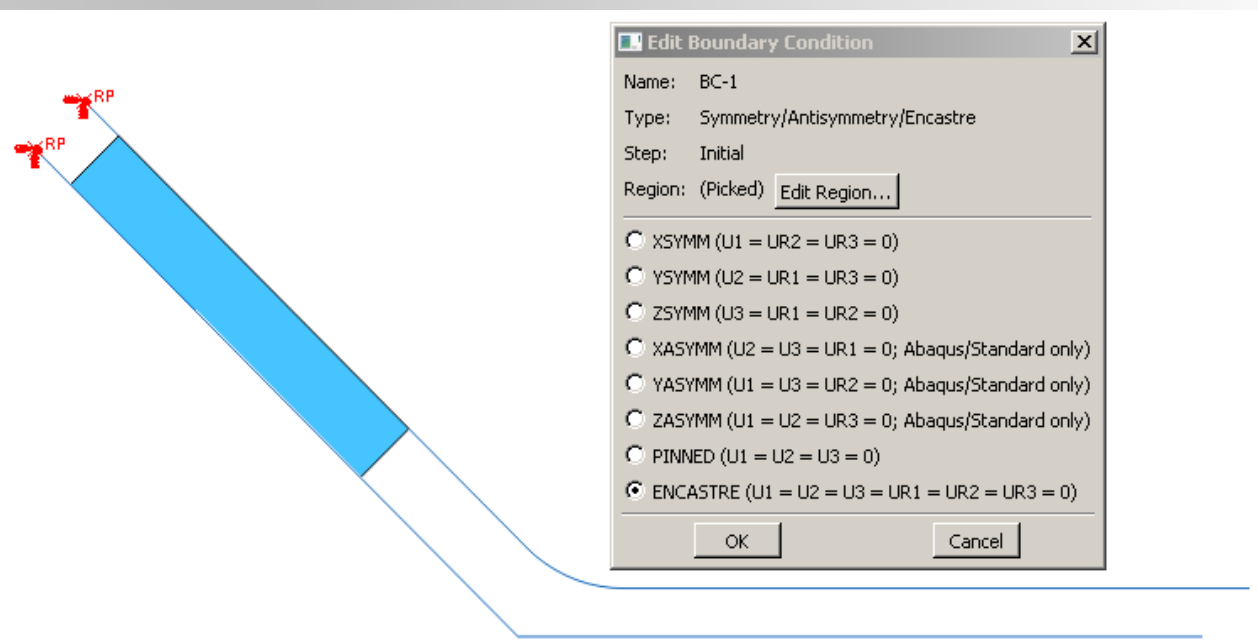
Acceleration/Angular acceleration

Connector displacement

Connector velocity

Connector acceleration

Continue... Cancel



Edit Boundary Condition

Name: BC-1

Type: Symmetry/Antisymmetry/Encastre

Step: Initial

Region: (Picked) Edit Region...

☐ XSYMM ($U_1 = U_2 = U_3 = 0$)

☐ YSYMM ($U_2 = U_1 = U_3 = 0$)

☐ ZSYMM ($U_3 = U_1 = U_2 = 0$)

☐ XASYMM ($U_2 = U_3 = U_1 = 0$; Abaqus/Standard only)

☐ YASYMM ($U_1 = U_3 = U_2 = 0$; Abaqus/Standard only)

☐ ZASYMM ($U_1 = U_2 = U_3 = 0$; Abaqus/Standard only)

☐ PINNED ($U_1 = U_2 = U_3 = 0$)

☒ ENCASTRE ($U_1 = U_2 = U_3 = U_1 = U_2 = U_3 = 0$)

OK Cancel

Create Load

Name: Load-1

Step: Step-1

Procedure: Dynamic, Explicit

Category

☒ Mechanical

☐ Thermal

☐ Acoustic

☐ Fluid

☐ Electrical

☐ Mass diffusion

☐ Other

Types for Selected Step

Concentrated force

Moment

Pressure

Shell edge load

Surface traction

Body force

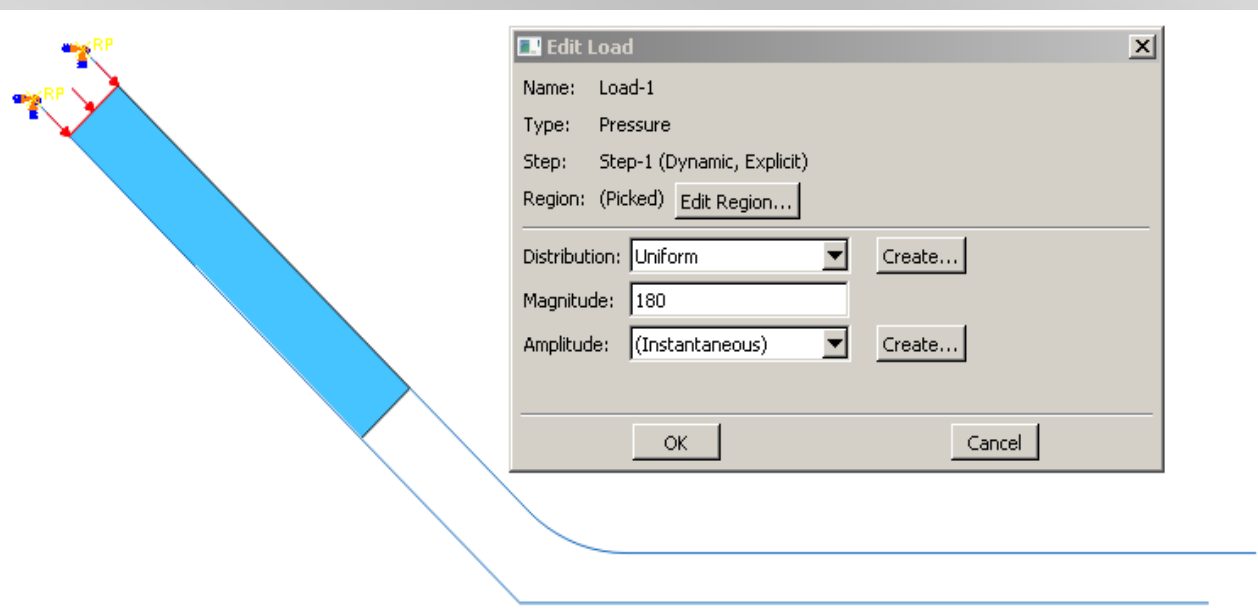
Line load

Gravity

Connector force

Connector moment

Continue... Cancel



Edit Load

Name: Load-1

Type: Pressure

Step: Step-1 (Dynamic, Explicit)

Region: (Picked) Edit Region...

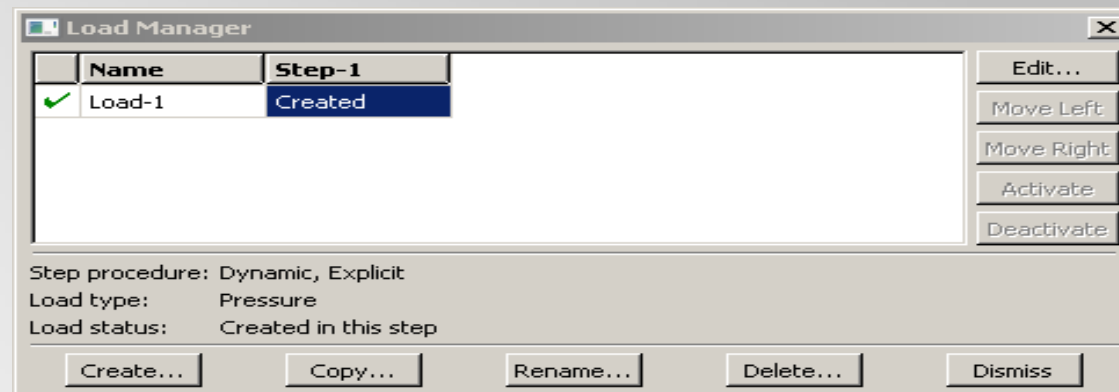
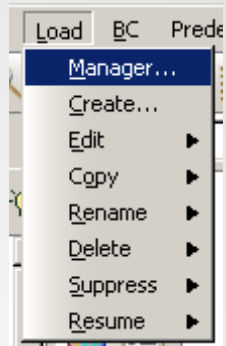
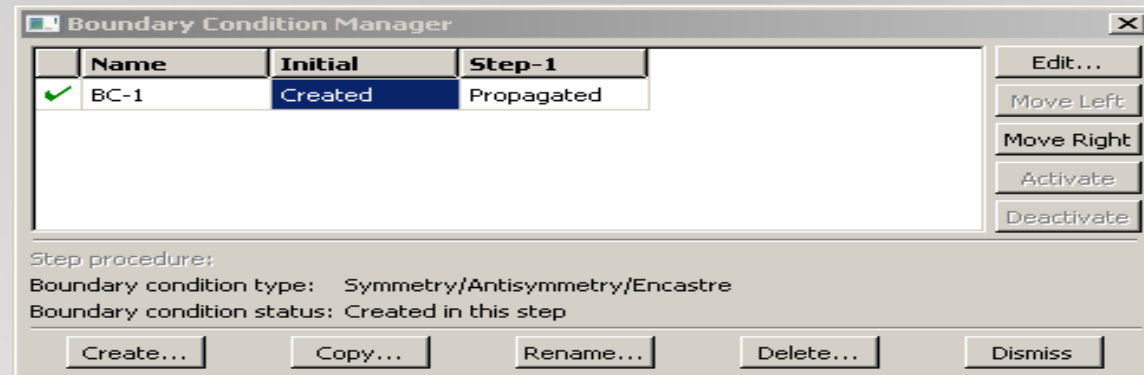
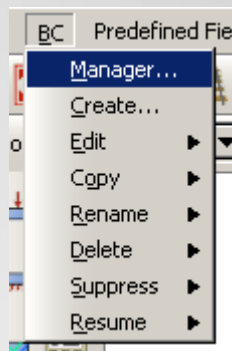
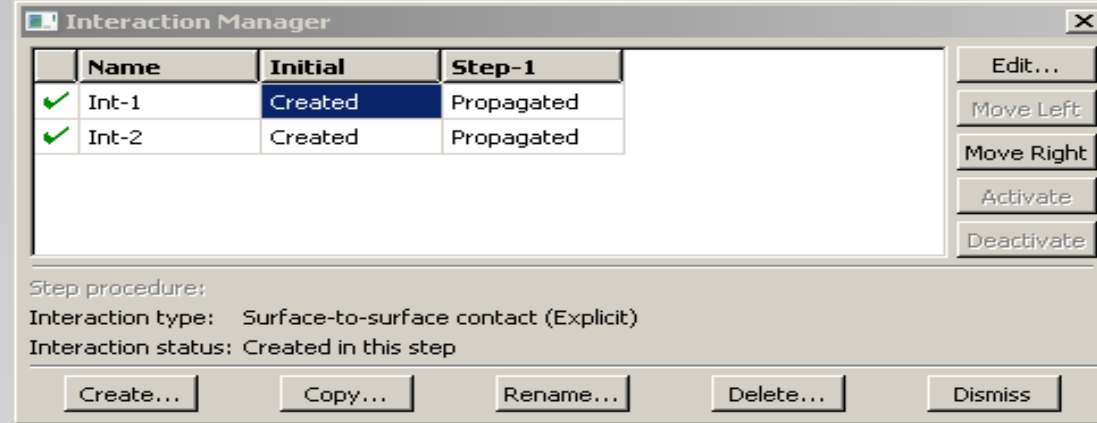
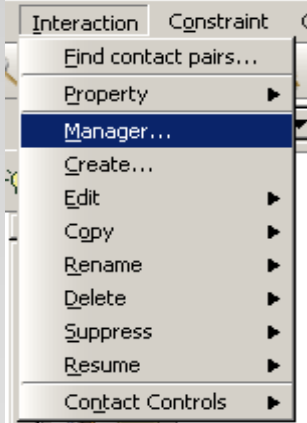
Distribution: Uniform Create...

Magnitude: 180

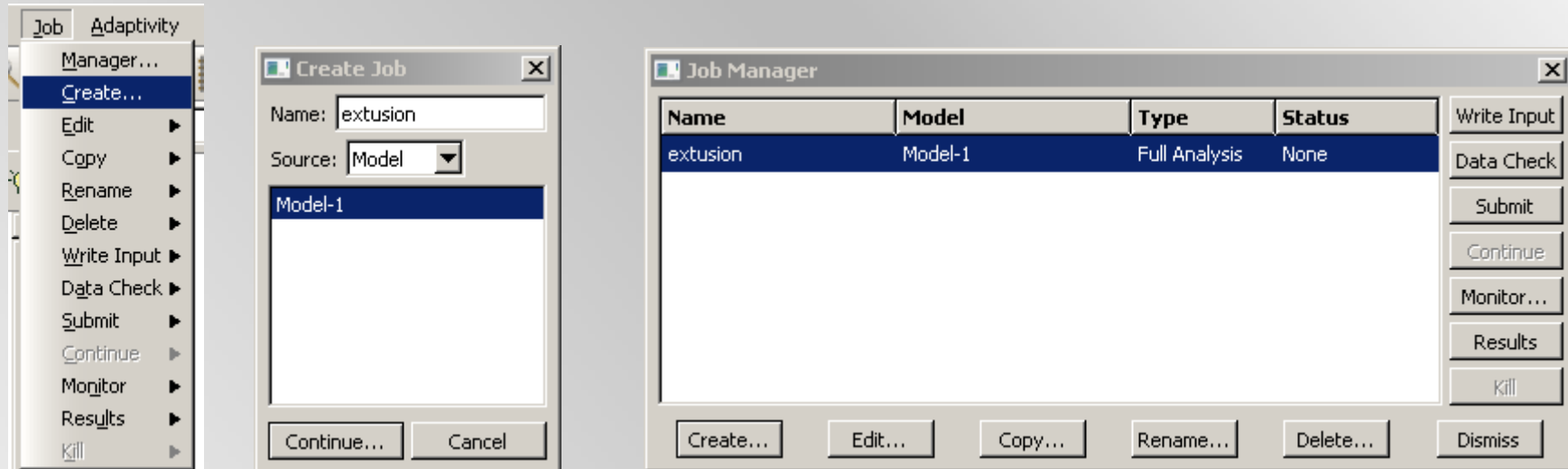
Amplitude: (Instantaneous) Create...

OK Cancel

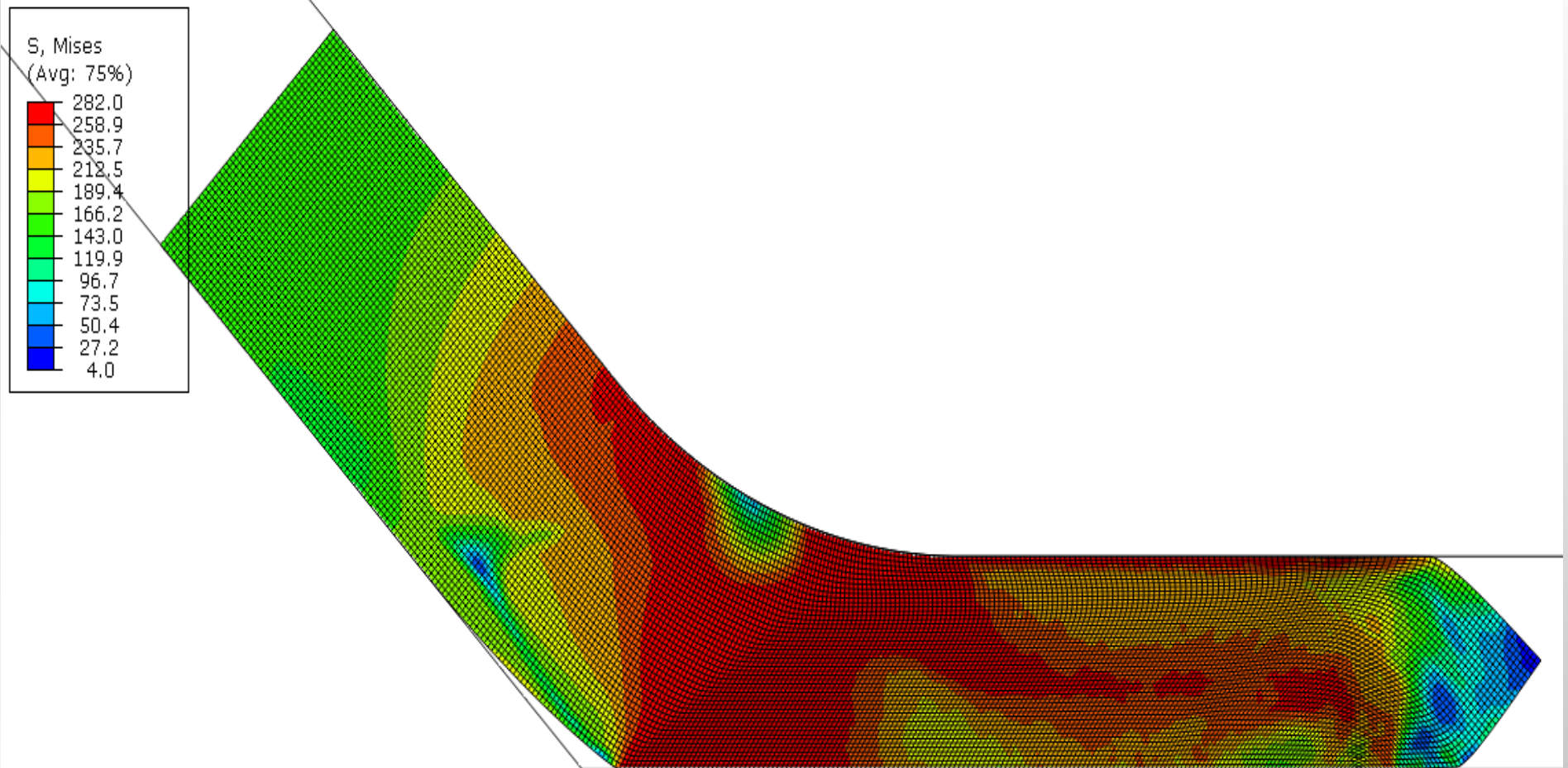
Interaction, Boundary condition and Load manager



Job module



Visualization module : Results

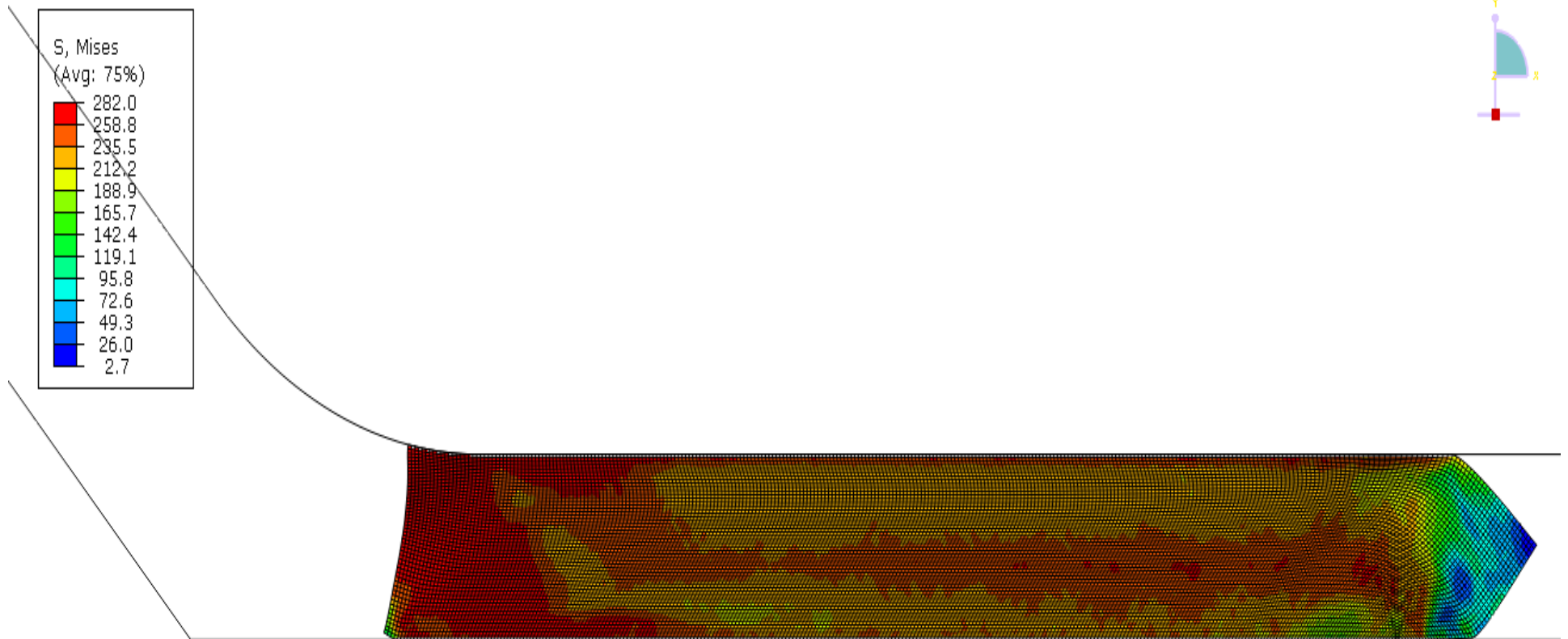


ODB: 14.odb Abaqus/Explicit Version 6.7-1 Sat Jan 28 20:02:41 Central Standard Time 2012



Step: Step-1
Increment 30147: Step Time = 9.7501E-03
Primary Var: S, Mises
Deformed Var: U Deformation Scale Factor: +1.0e+00

Visualization module: Results

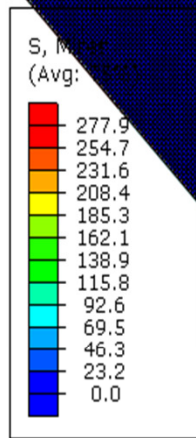


ODB: 14.odb Abaqus/Explicit Version 6.7-1 Sat Jan 28 20:02:41 Central Standard Time 2012



Step: Step-1
Increment 46253; Step Time = 1.2750E-02
Primary Var: S, Mises
Deformed Var: U Deformation Scale Factor: +1.0e+00

Visualization module: Animation



Step: Step-1 Frame: 0



ODB: 14.odb Abaqus/Explicit Version 6.7-1 Sat Jan 28 20:02:41 Central Standard Time 2012



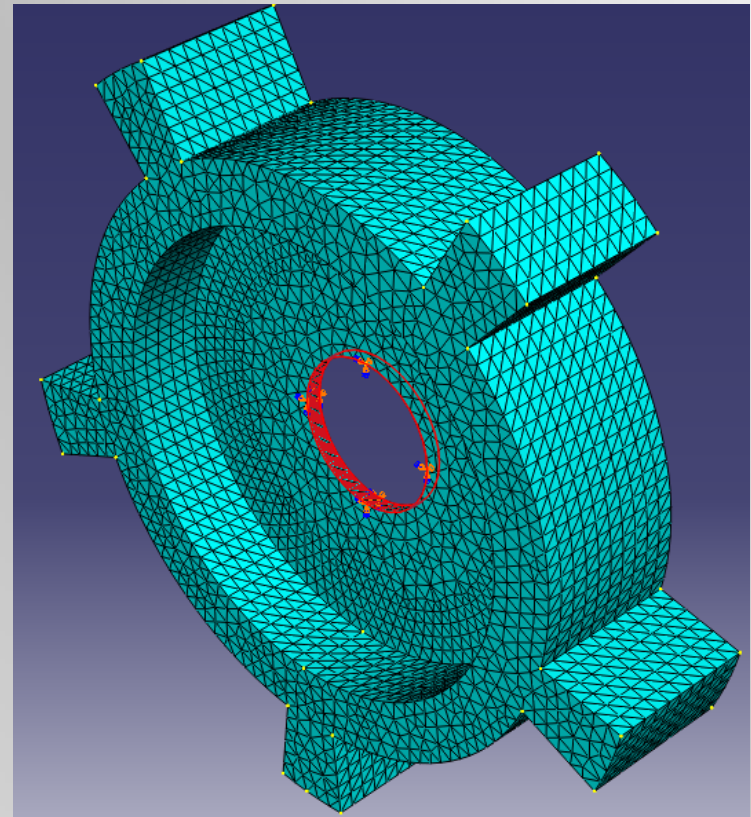
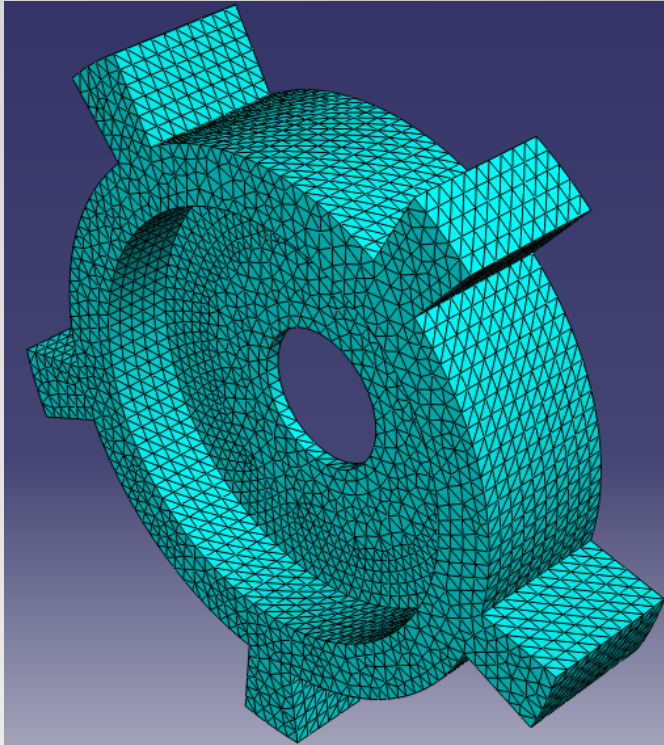
Step: Step-1
Increment 0: Step Time = 0.0
Primary Var: S, Mises
Deformed Var: U Deformation Scale Factor: +1.0e+00



Example 4: Frequency Analysis of Gear

Refer: Section 2.5.1 Abaqus Theory manual

Problem description



Part Module

Create Part

Name:

Modeling Space

☒ 3D ☐ 2D Planar ☐ Axisymmetric

Type

☒ Deformable
☐ Discrete rigid
☐ Analytical rigid

Options

None available

Base Feature

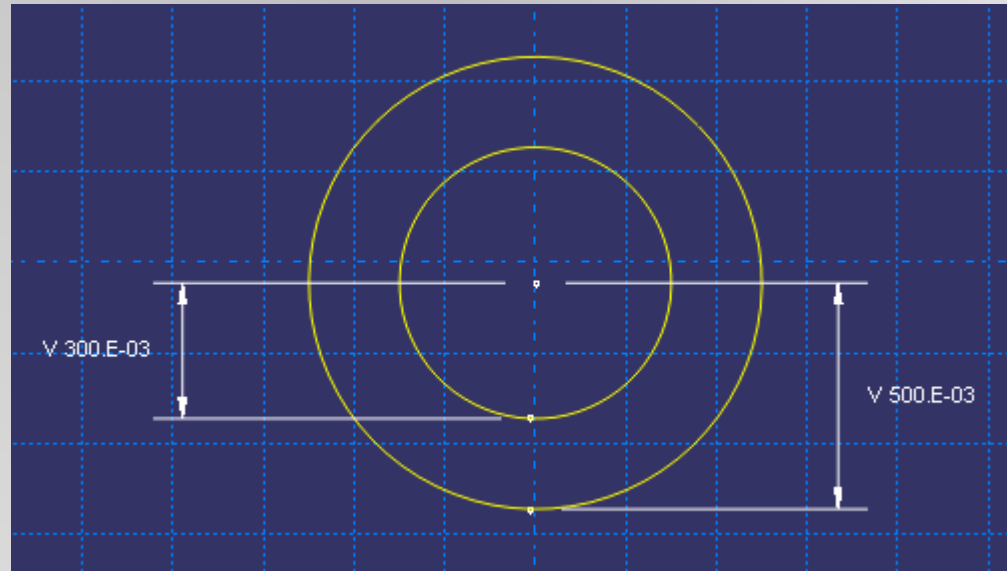
Shape

☒ Solid
☐ Shell
☐ Wire
☐ Point

Type

Extrusion
Revolution
Sweep

Approximate size:



Edit Base Extrusion

End Condition

Type: Blind

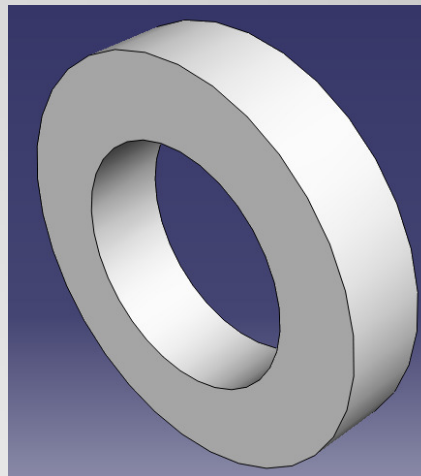
Depth:

Options

Note: Twist and draft cannot be specified together.

☐ Include twist, pitch: (Dist/Rev)

☐ Include draft, angle: (Degrees)



Tools Plug-ins Help

Query...
Reference Point...
Set
Surface
Partition...
Datum...
Geometry Repair...
Display Group
Customize...

Create Datum

Type

☐ Point ☒ Axis ☐ Plane ☐ CSYS

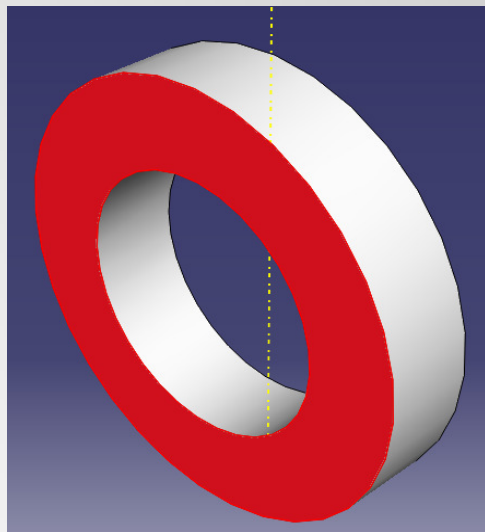
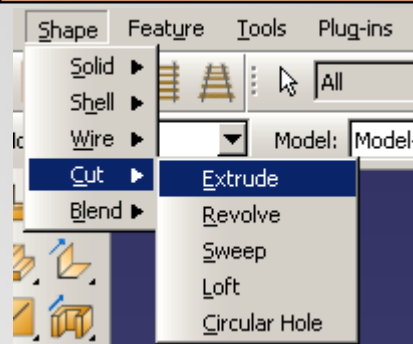
Method

Principal axis
Intersection of 2 planes
Straight edge
2 points
Axis of cylinder
Normal to plane, thru point
Parallel to line, thru point
3 points on circle
Rotate from line

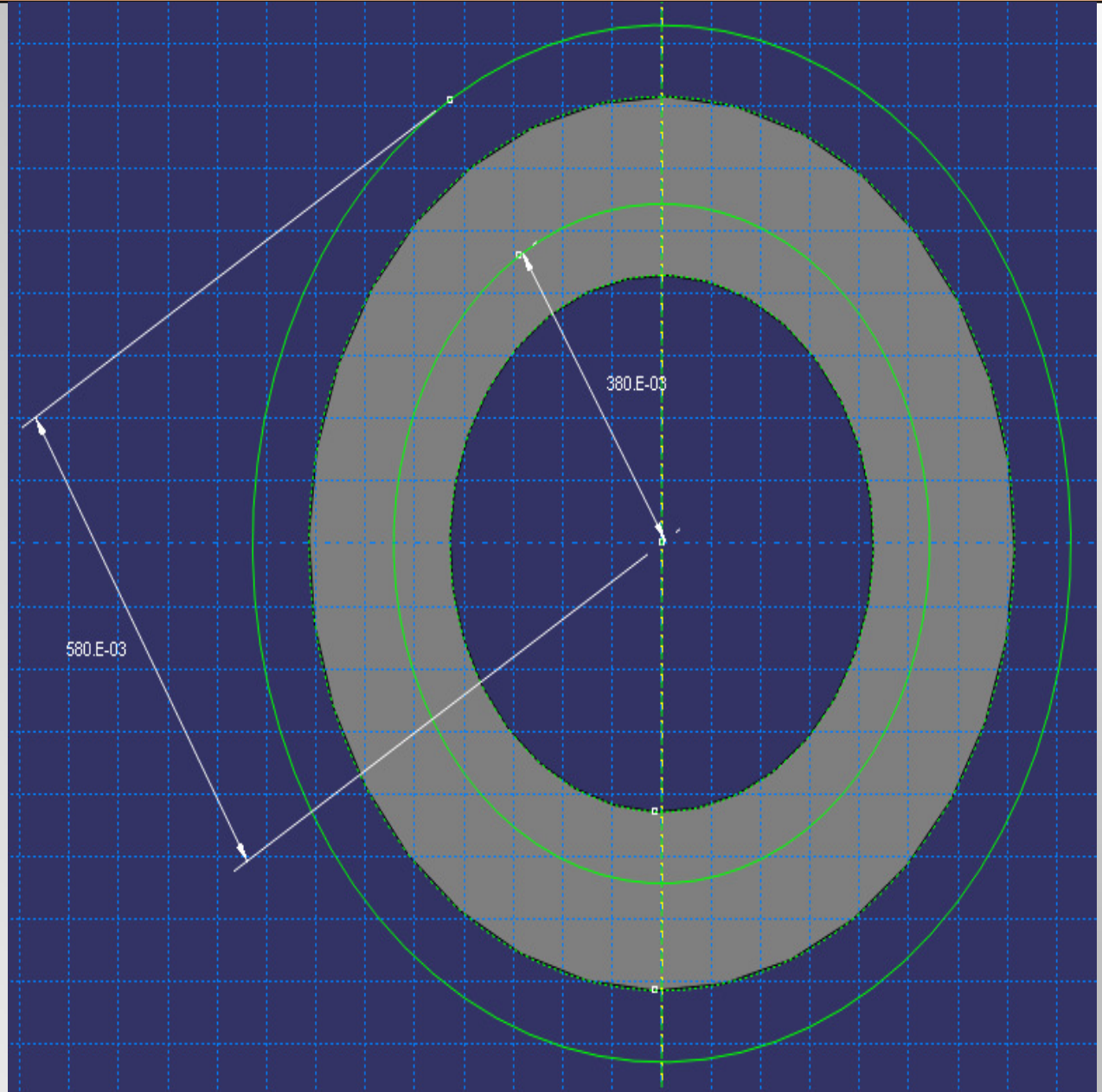
Principal axis choice:

Choose Y-axis

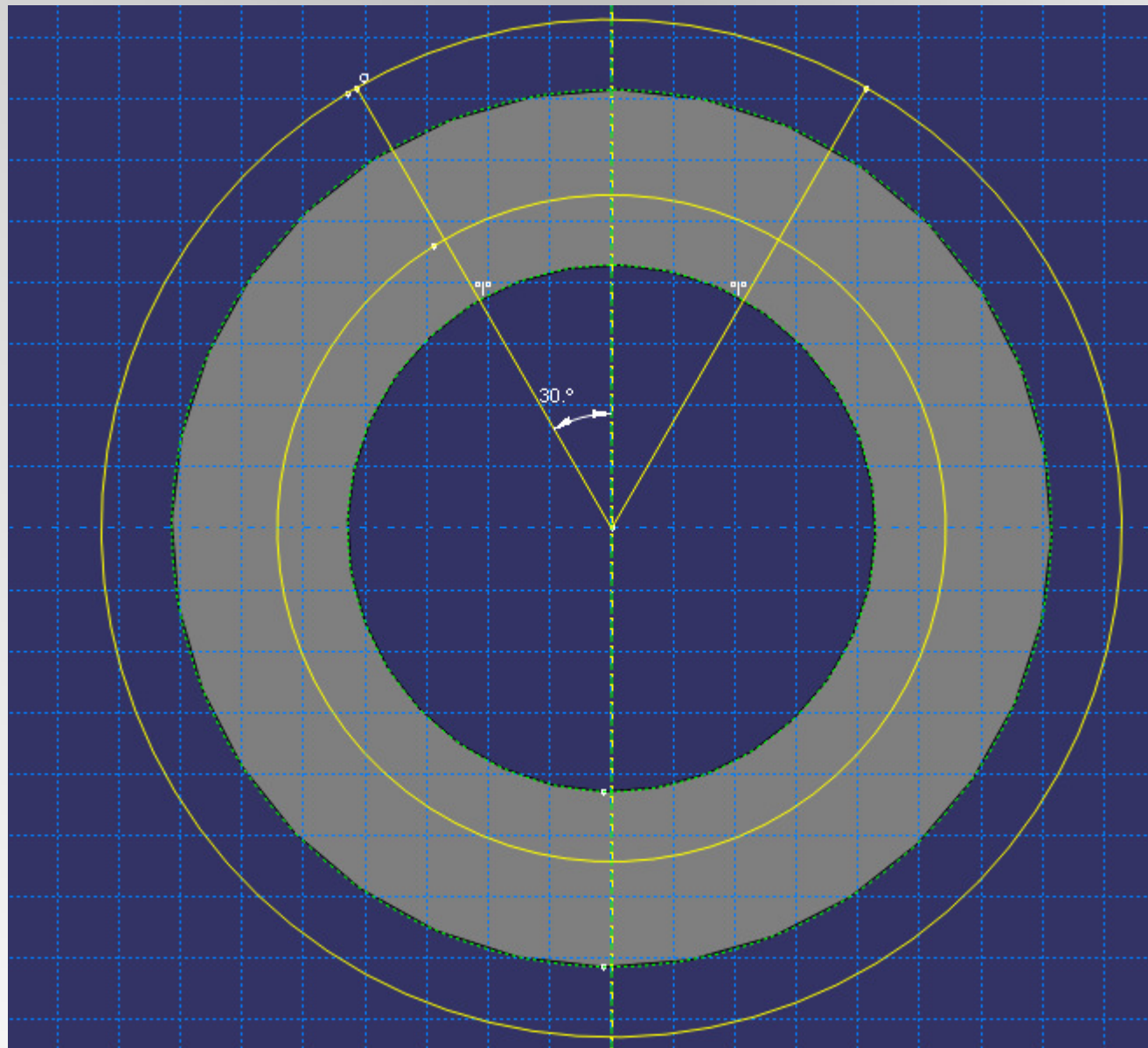
Part Module



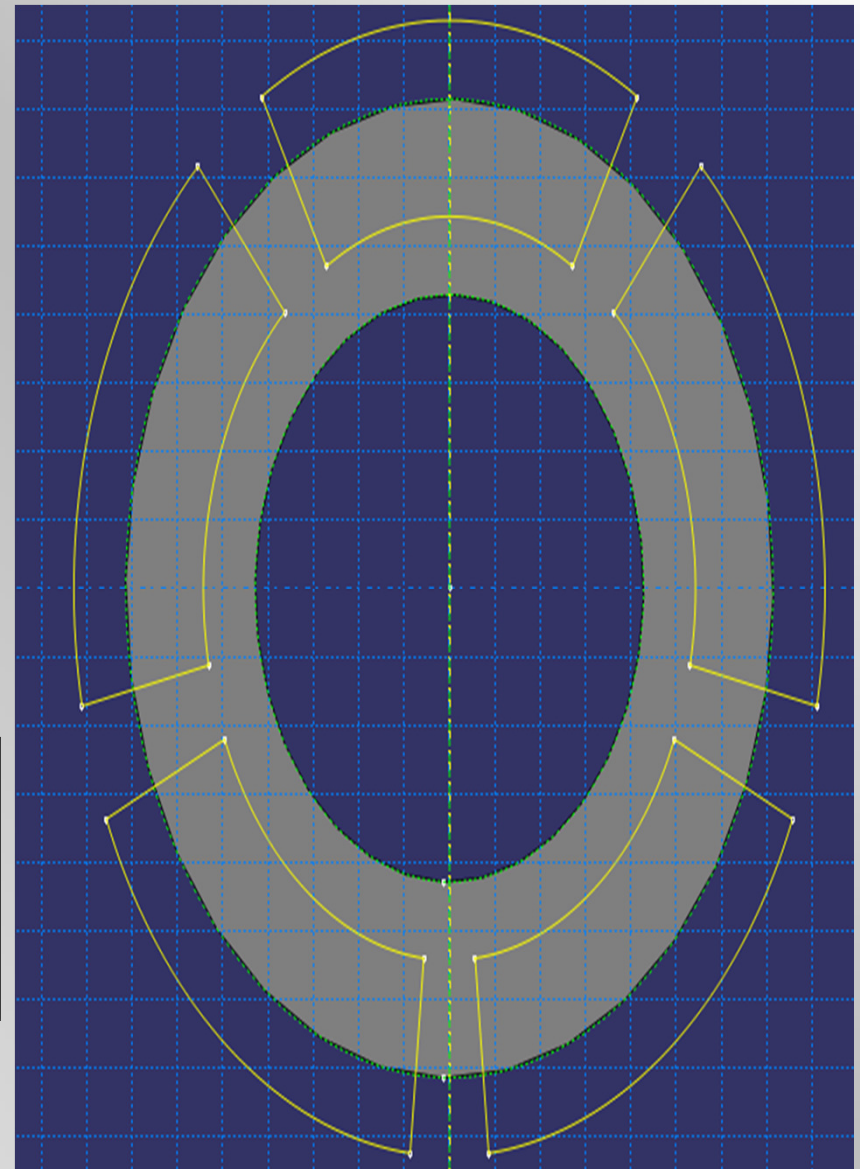
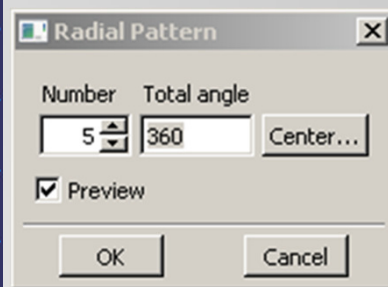
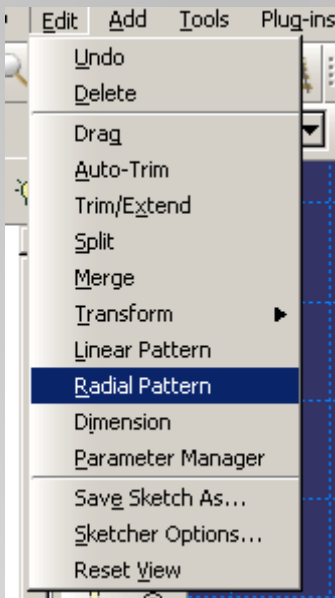
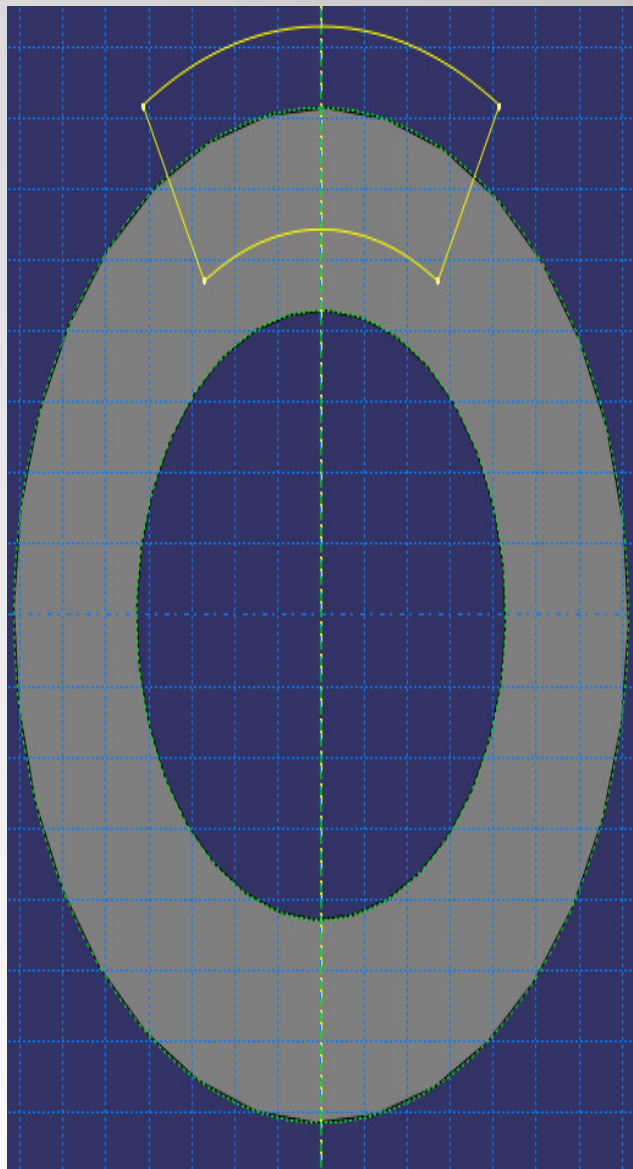
When asked to choose edge or axis, choose the datum axis created in last slide



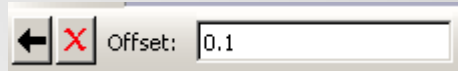
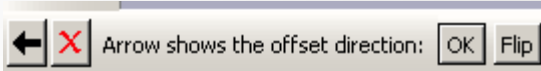
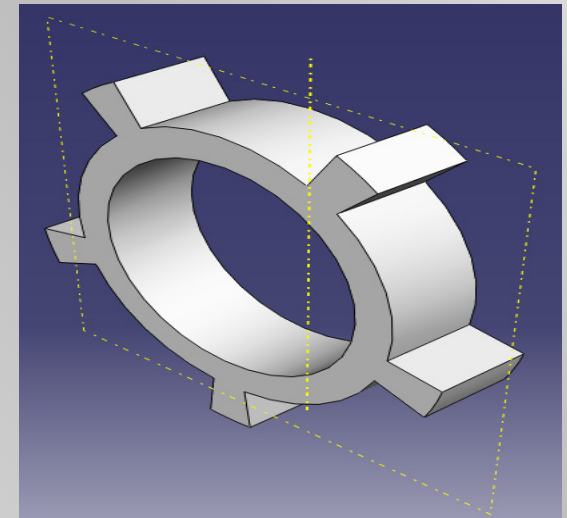
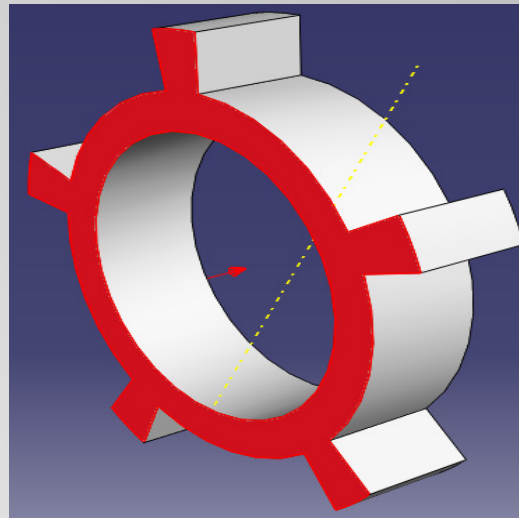
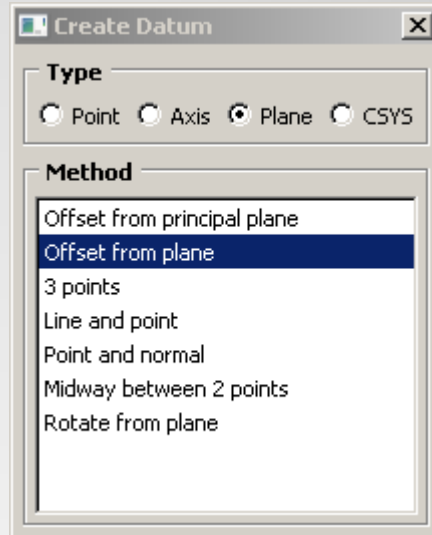
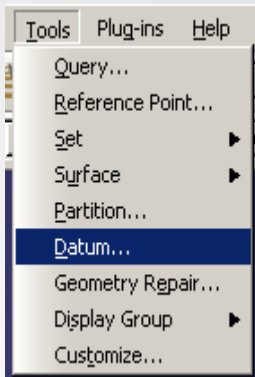
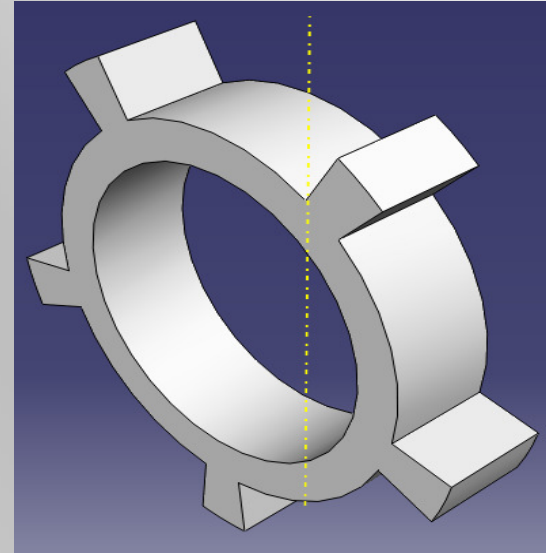
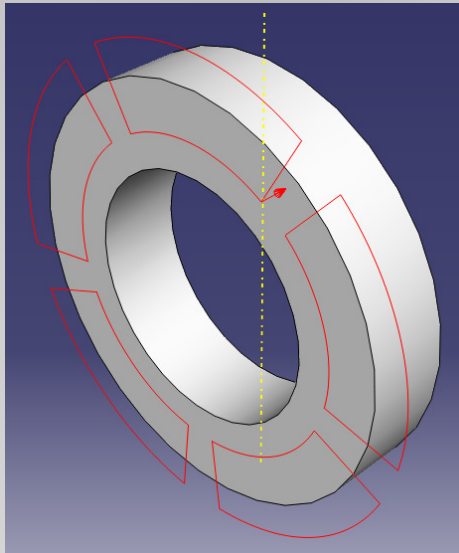
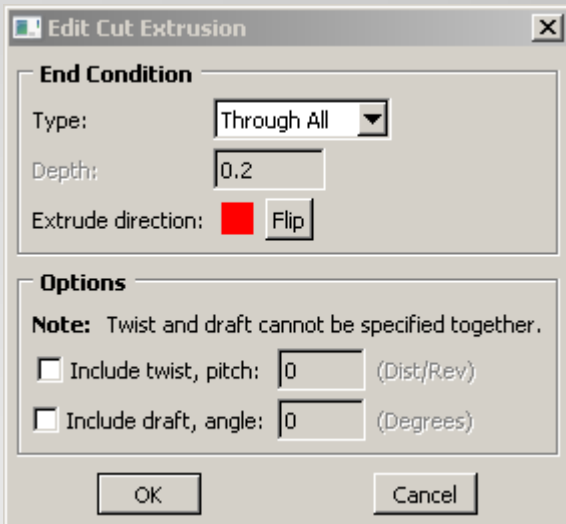
Part Module



Part Module

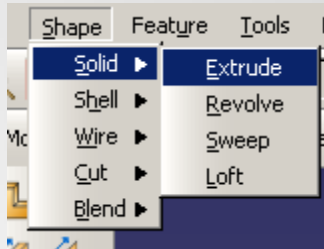


Part Module

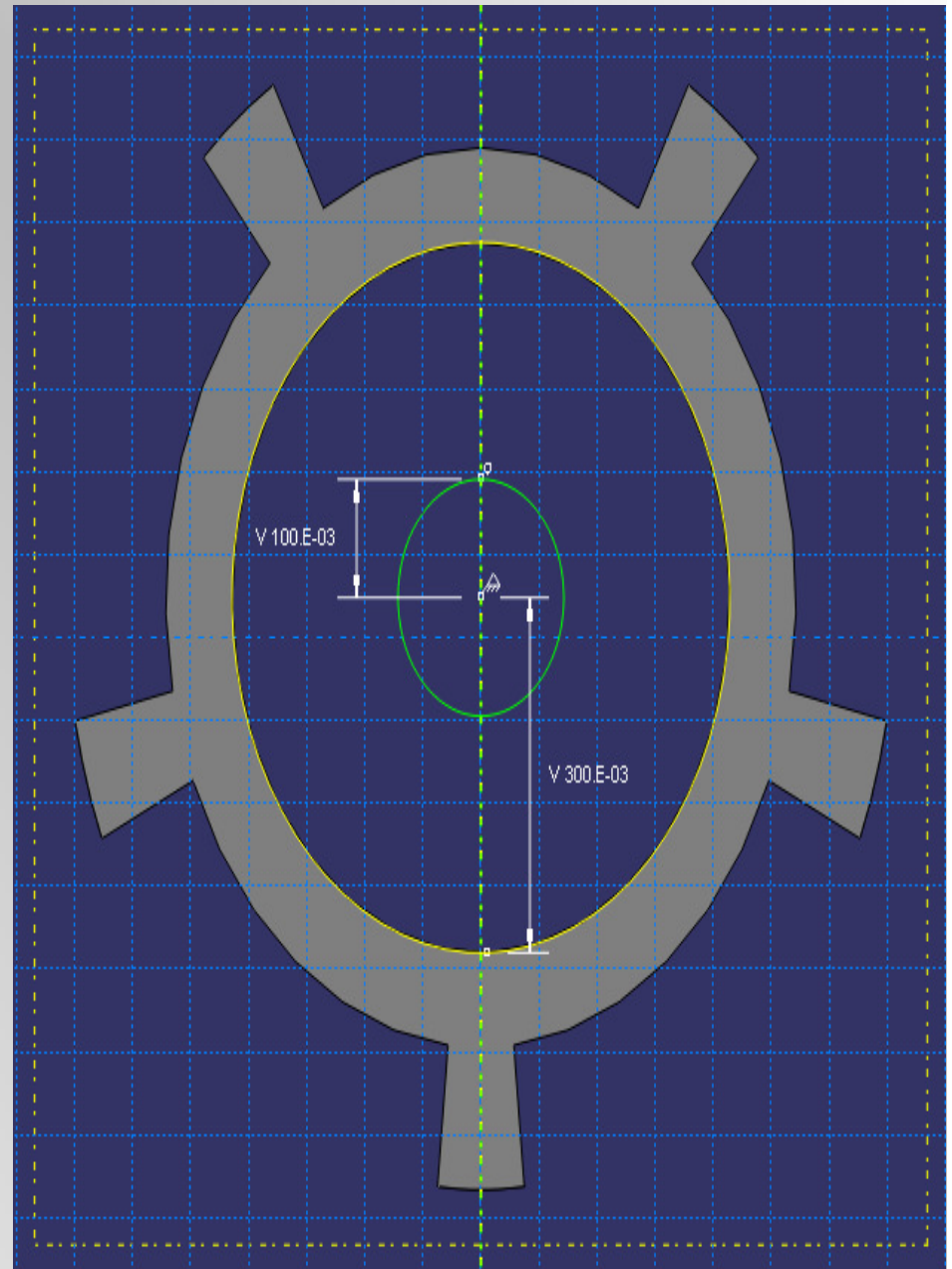
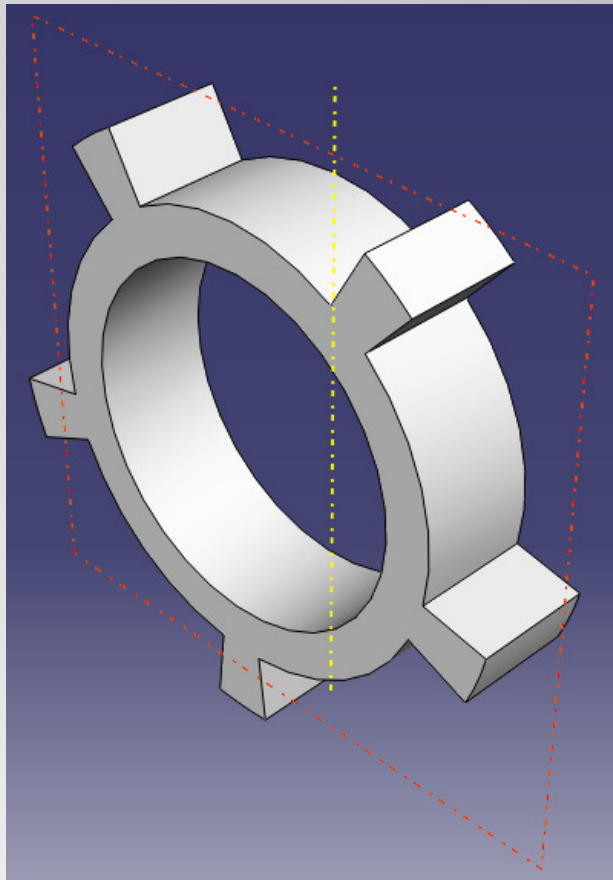


Purpose of creating datum plane will be explained in next slide

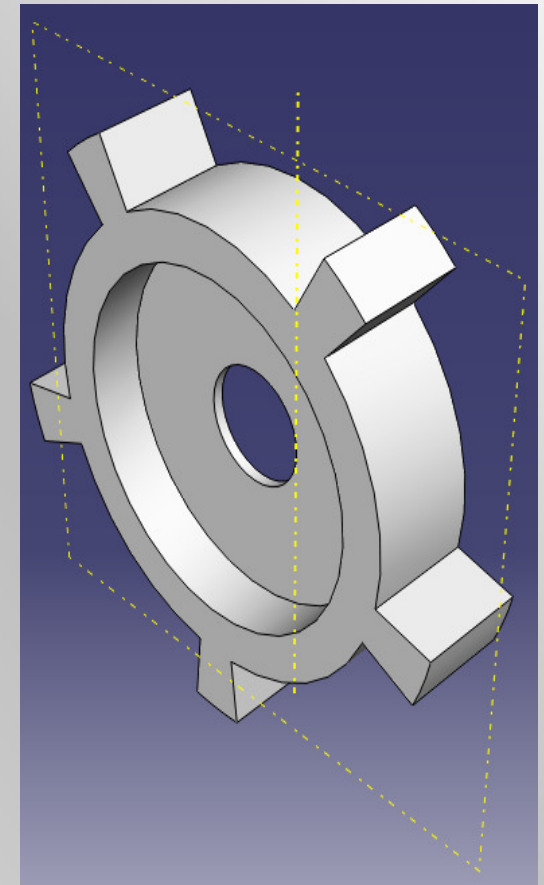
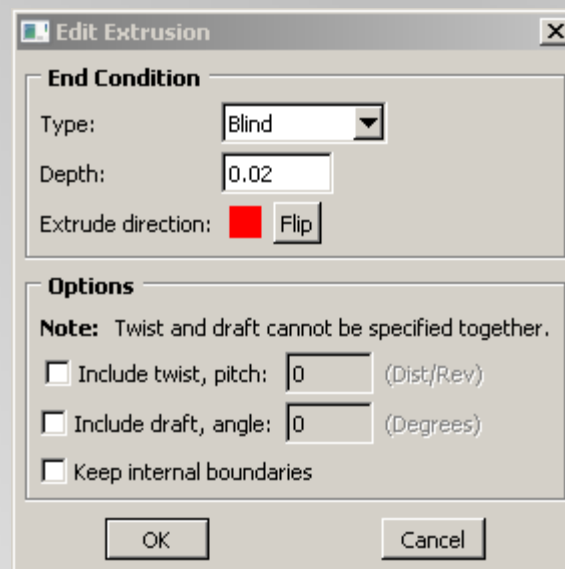
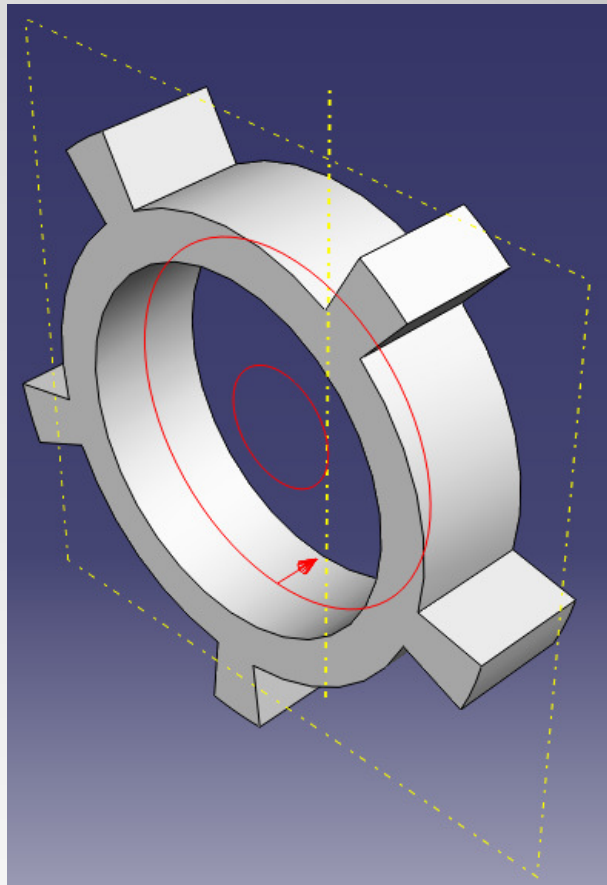
Part Module



When asked to select a plane for extrusion, select the datum plane as shown in the picture (the datum plane was created in the last slide for this purpose)



Part Module



Property Module: Material Properties

Edit Material

Name: Aluminum

Description:

Material Behaviors

Density

Elastic

General Mechanical Thermal Other Delete

Elastic

Type: Isotropic Suboptions

☐ Use temperature-dependent data

Number of field variables: 0

Moduli time scale (for viscoelasticity): Long-term

☐ No compression

☐ No tension

Data

	Young's Modulus	Poisson's Ratio
1	65475e6	0.336

OK Cancel

Create Section

Name: Section-1

Category **Type**

☒ Solid ☒ Homogeneous

☐ Shell ☐ Generalized plane strain

☐ Beam

☐ Other

Continue... Cancel

Edit Section

Name: Section-1

Type: Solid, Homogeneous

Material: Aluminum Create...

Plane stress/strain thickness: 1

OK Cancel

Section Profile Composite

Manager...

Create...

Edit

Copy

Rename

Delete

Assignment Manager...

Section Assignment Manager

Section Name (Type)	Material Name	Region
Section-1 (Solid, Homogeneous)	Aluminum	(Picked)

Create... Edit... Delete... Dismiss

Assembly & Step module

Create Instance

Parts

Part-1

Instance Type

☒ Dependent (mesh on part)
☐ Independent (mesh on instance)

Note: To change a Dependent instance's mesh, you must edit its part's mesh.

☐ Auto-offset from other instances

OK Apply Cancel

Create Step

Name: Step-1

Insert new step after

Initial

Procedure type: Linear perturbation

Buckle
Frequency
Static, Linear perturbation
Steady-state dynamics, Direct

Continue... Cancel

Edit Step

Name: Step-1

Type: Frequency

Basic Parallel Lanczos Other

Description:

Nlgeom: Off

Eigsolver: ☒ Lanczos ☐ Subspace ☐ AMS

Number of eigenvalues requested: ☐ All in frequency range
☒ Value: 100

☐ Frequency shift (cycles/time)**2:

☐ Minimum frequency of interest (cycles/time):

☐ Maximum frequency of interest (cycles/time):

☒ Include acoustic-structural coupling where applicable

Block size: ☒ Default ☐ Value:

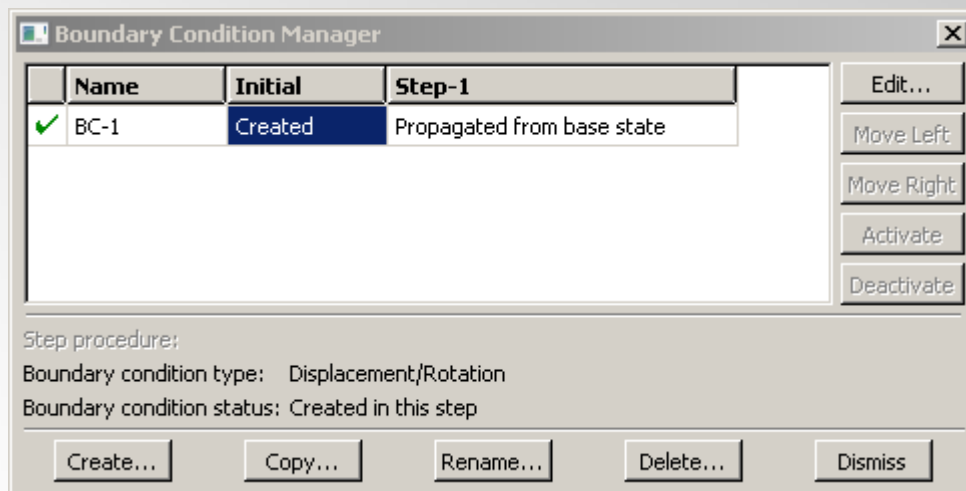
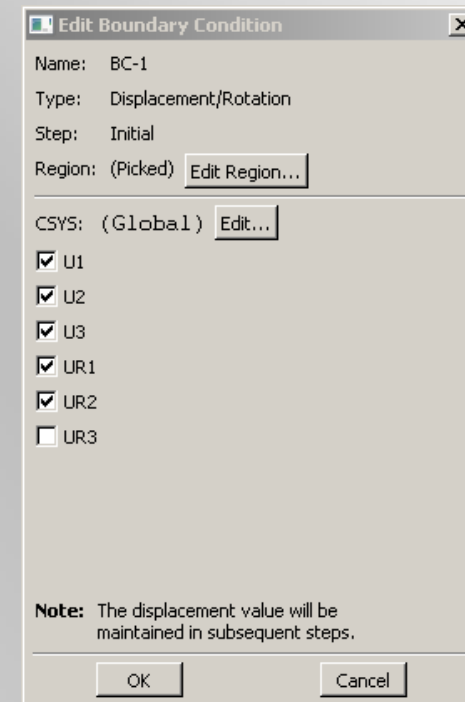
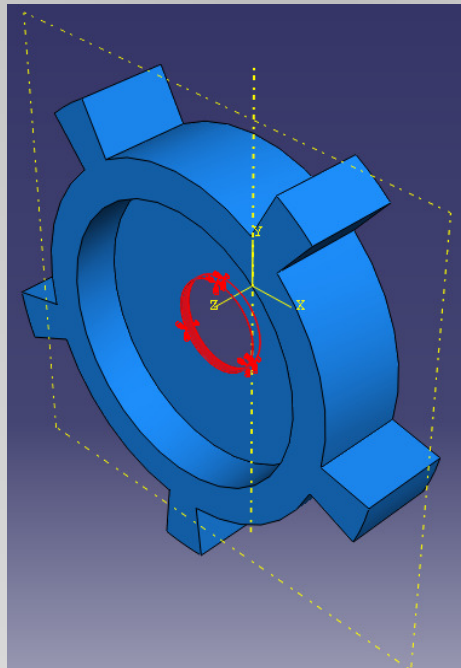
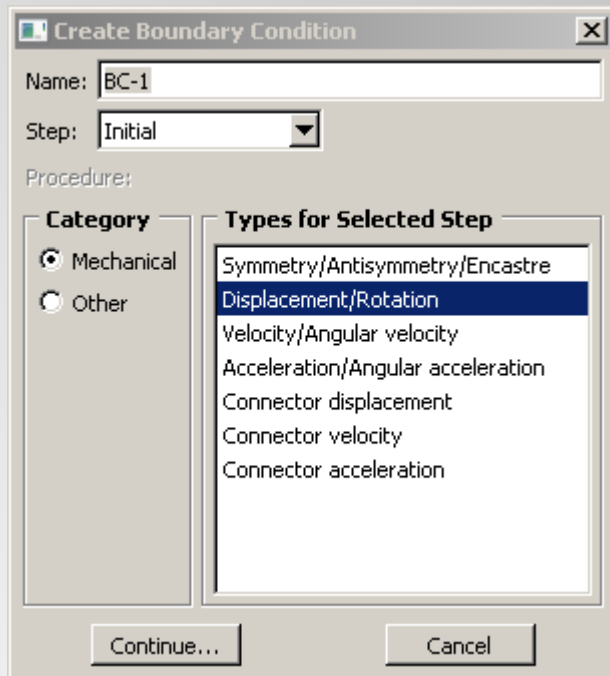
Maximum number of block Lanczos steps: ☒ Default ☐ Value:

☐ Use SIM-based linear dynamics procedures

☐ Include residual modes

OK Cancel

Load module : Boundary Conditions

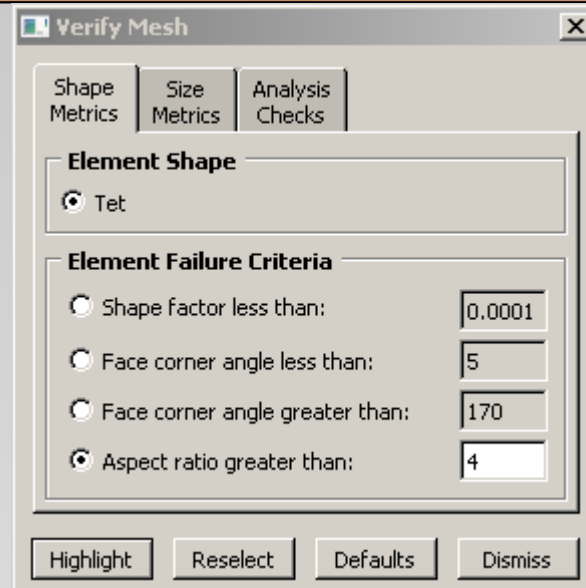
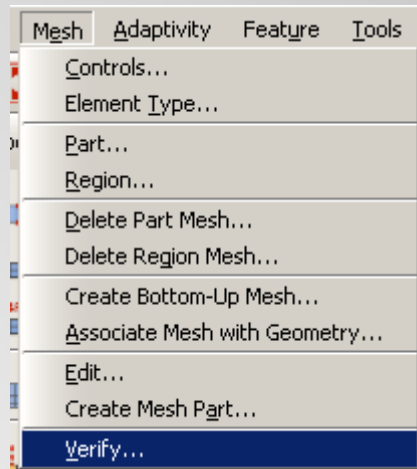


Mesh Module

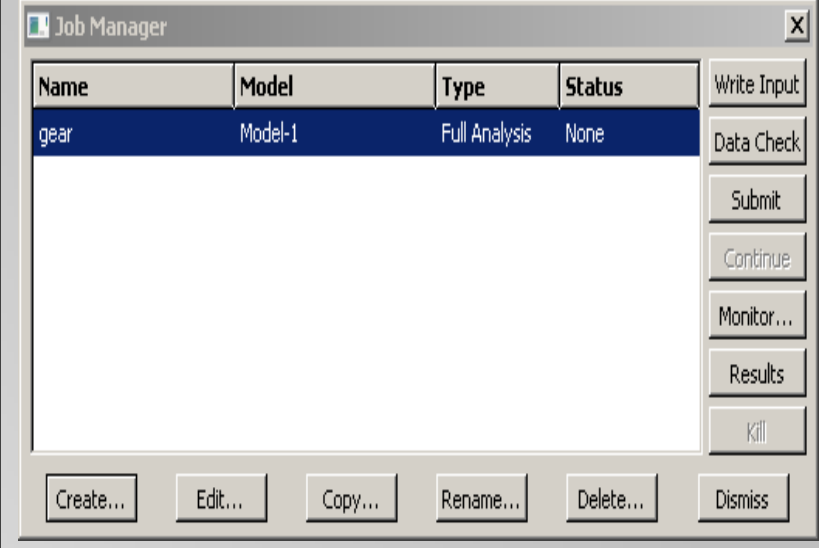
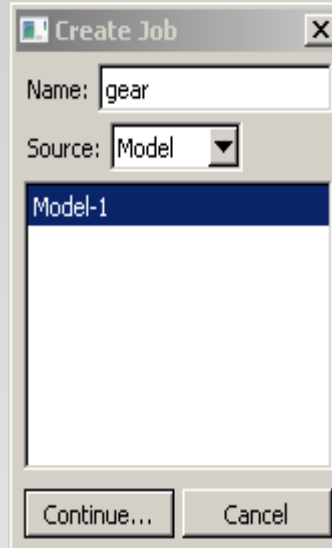
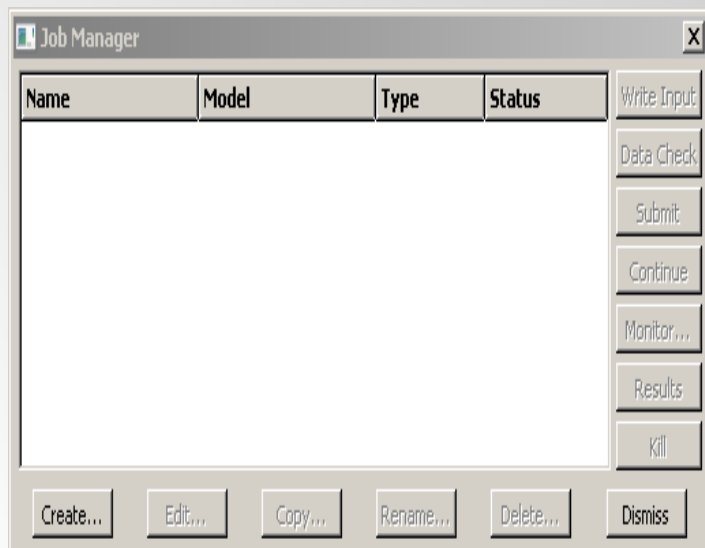
The screenshot displays the Mesh Module software interface with several key components:

- Main Menu:** View, Seed, Mesh, Adaptivity, F.
- Mesh Menu:**
 - Part...
 - Edge By Number...
 - Edge By Size... (highlighted)
 - Edge Biased...
 - Delete Part Seeds...
 - Delete Edge Seeds...
- Mesh Controls Dialog:**
 - Element Shape:** Hex, Hex-dominated, Tet (selected), Wedge.
 - Technique:** As is, Free (selected), Structured, Sweep, Bottom-up, Multiple.
 - Algorithm:**
 - ☒ Use default algorithm
 - ☐ Increase size of interior elements
 - Moderate growth
 - Maximum growth
 - ☒ Use mapped tri meshing on bounding faces where appropriate
- Mesh Menu (Secondary):**
 - Controls...
 - Element Type...
 - Part... (highlighted)
 - Region...
 - Delete Part Mesh...
 - Delete Region Mesh...
 - Create Bottom-Up Mesh...
 - Associate Mesh with Geometry...
 - Edit...
 - Create Mesh Part...
 - Verify...
- Element Type Dialog:**
 - Element Library:** Standard (selected), Explicit.
 - Geometric Order:** Linear (selected), Quadratic.
 - Family:** 3D Stress (selected), Acoustic, Cohesive, Continuum Shell.
 - Hex | Wedge | Tet:** Tet (selected).
 - Element Controls:**
 - ☐ Hybrid formulation
 - Distortion control: ☒ Use default, ☐ Yes, ☐ No
 - Length ratio: 0.1
 - Linear bulk viscosity scaling factor: 1
 - Quadratic bulk viscosity scaling factor: 1
 - Note:** To select an element shape for meshing, select "Mesh->Controls" from the main menu bar.
- Meshed Model:** A 3D model of a gear-like part with a central hole, fully meshed with a cyan-colored tetrahedral mesh.

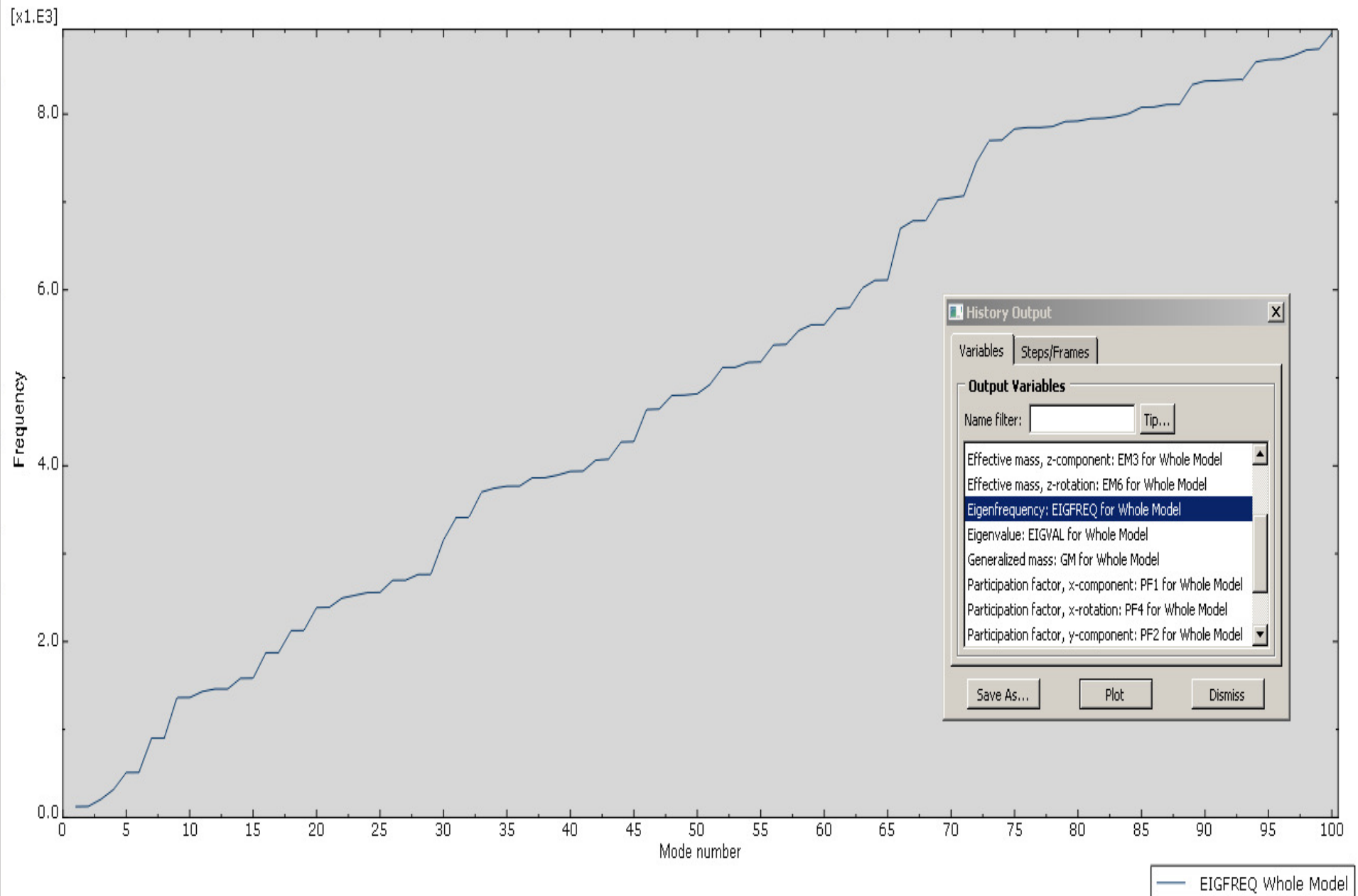
Job Module & Mesh module



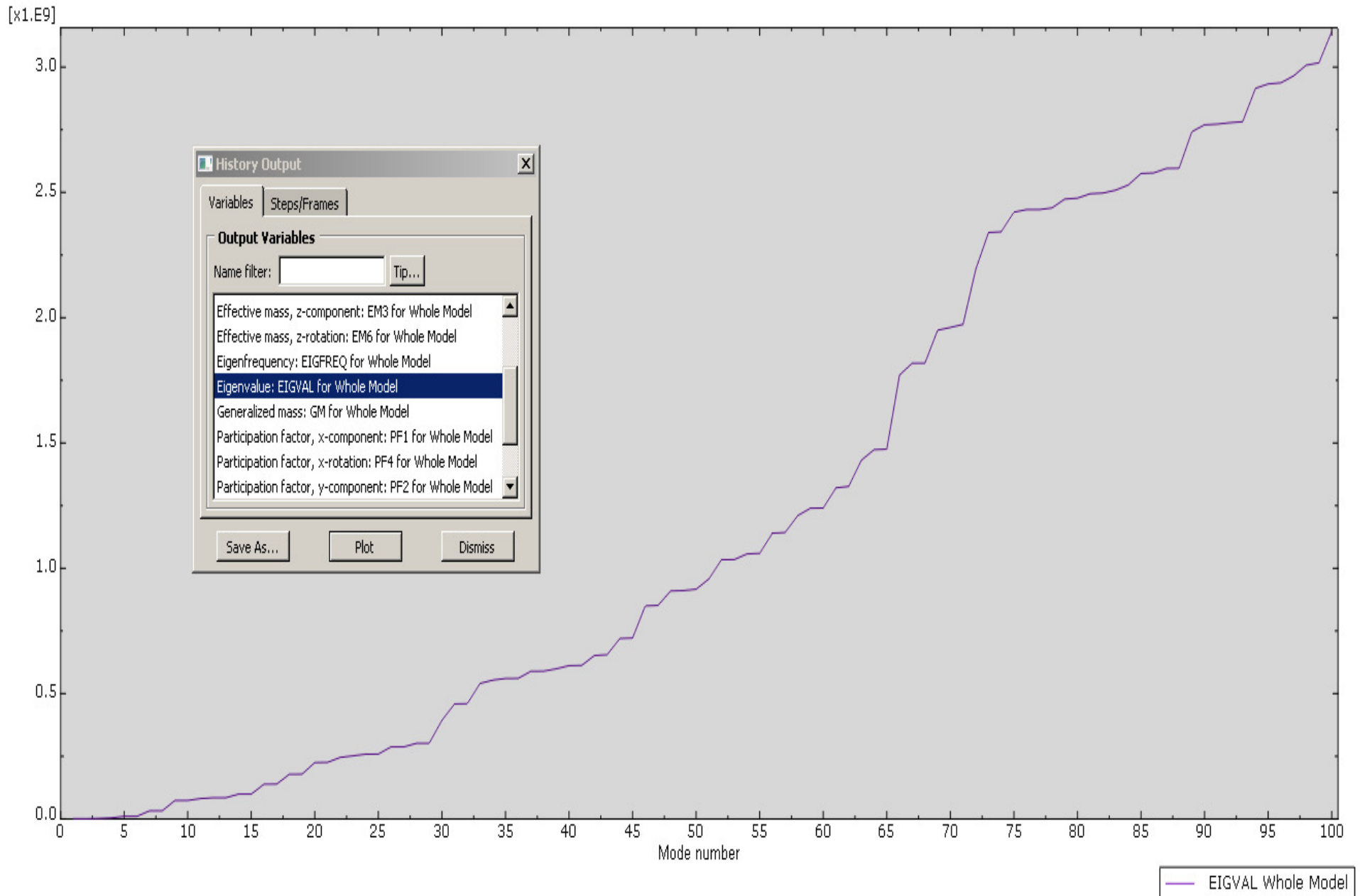
Part: Part-1
 Tet elements: 44226, Aspect ratio > 4: 0 (0%)
 Average aspect ratio: 1.59, Worst aspect ratio: 2.83



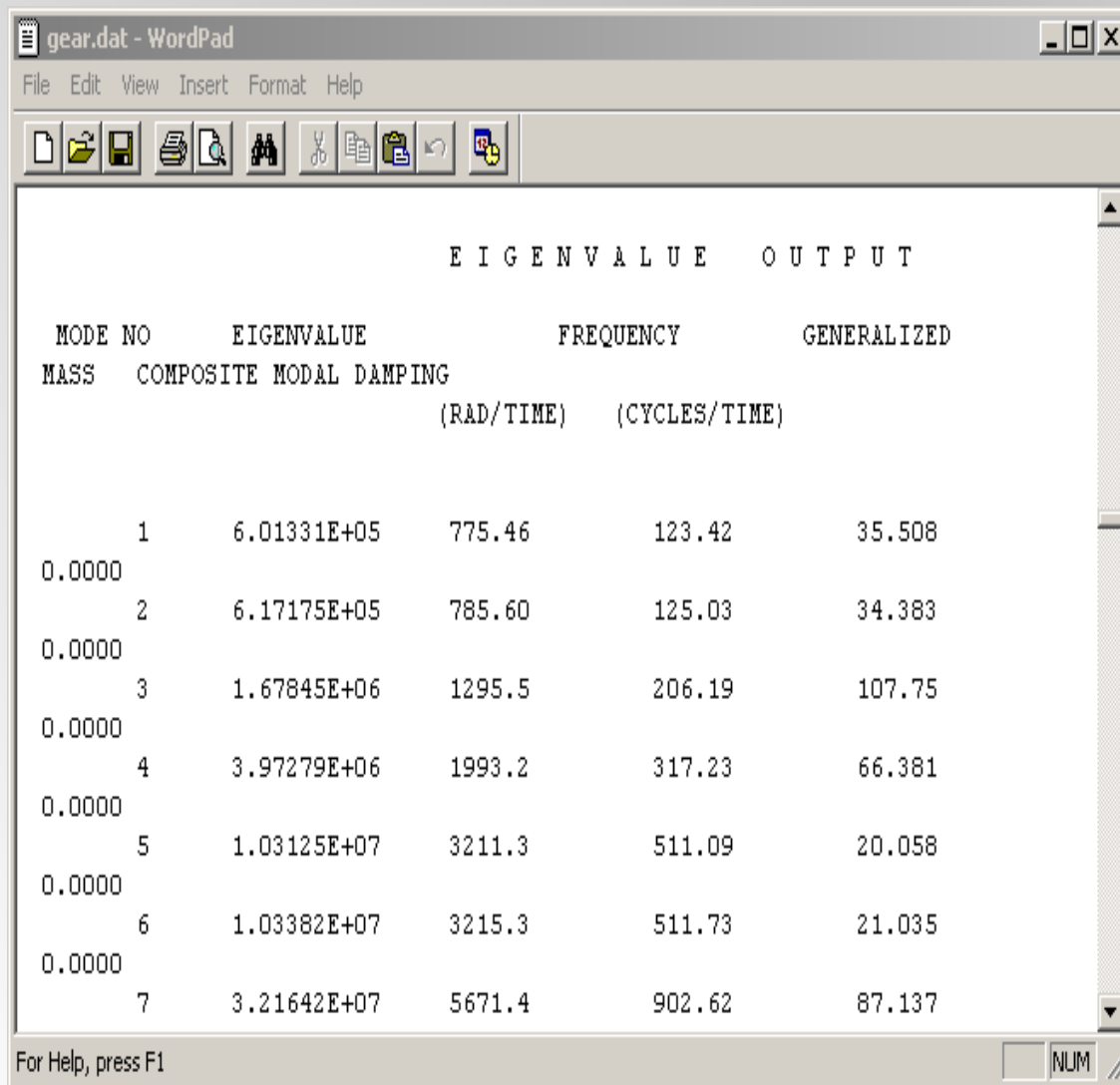
Visualization: Plot History output



Visualization: Plot History output



Results: Dat file



gear.dat - WordPad

File Edit View Insert Format Help

E I G E N V A L U E O U T P U T

MODE NO	EIGENVALUE	FREQUENCY	GENERALIZED
MASS	COMPOSITE MODAL DAMPING	(RAD/TIME)	(CYCLES/TIME)
1	6.01331E+05	775.46	123.42
0.0000			35.508
2	6.17175E+05	785.60	125.03
0.0000			34.383
3	1.67845E+06	1295.5	206.19
0.0000			107.75
4	3.97279E+06	1993.2	317.23
0.0000			66.381
5	1.03125E+07	3211.3	511.09
0.0000			20.058
6	1.03382E+07	3215.3	511.73
0.0000			21.035
7	3.21642E+07	5671.4	902.62
			87.137

For Help, press F1

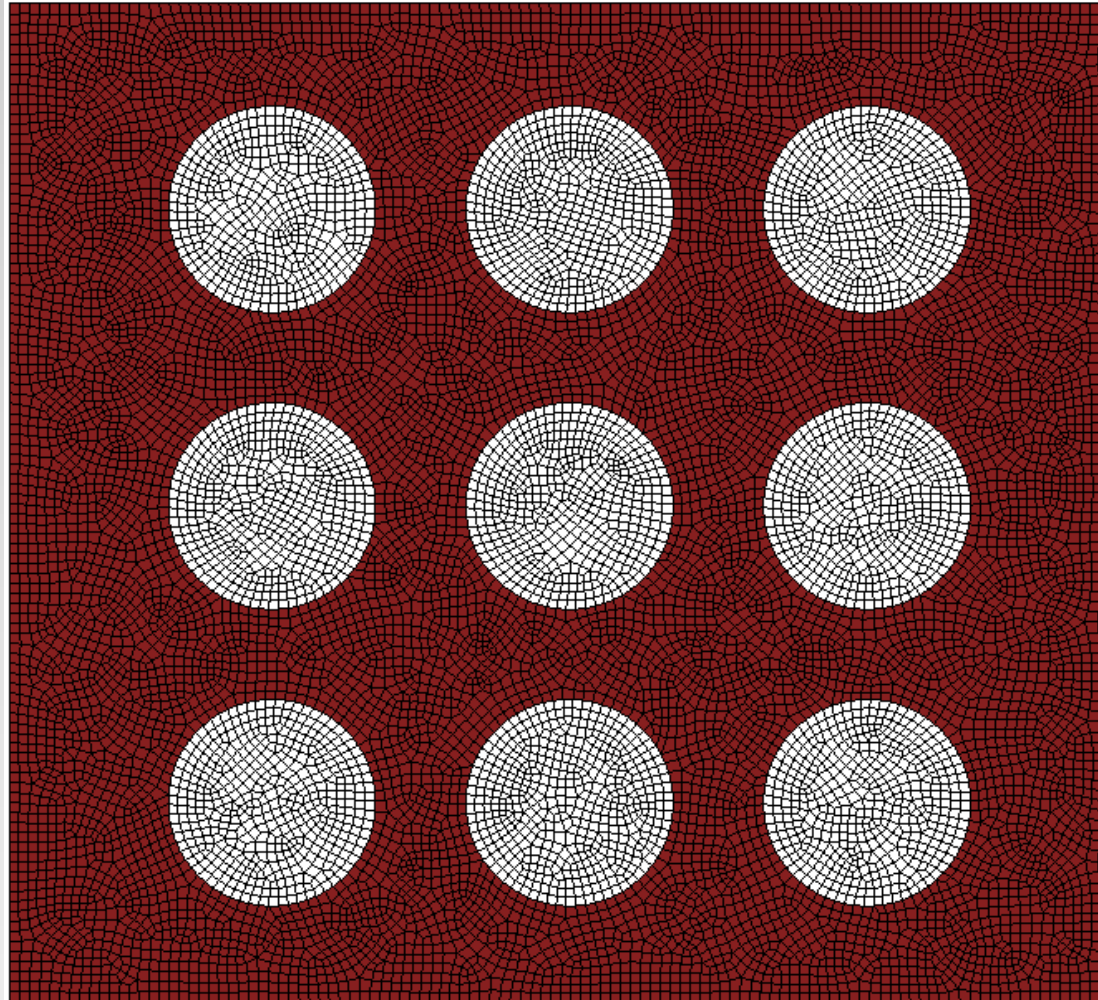
NUM

Refer: sections 7.5 and 7.6 of “Getting started with Abaqus interactive edition” manual >> for a more detailed post-processing of a similar problem

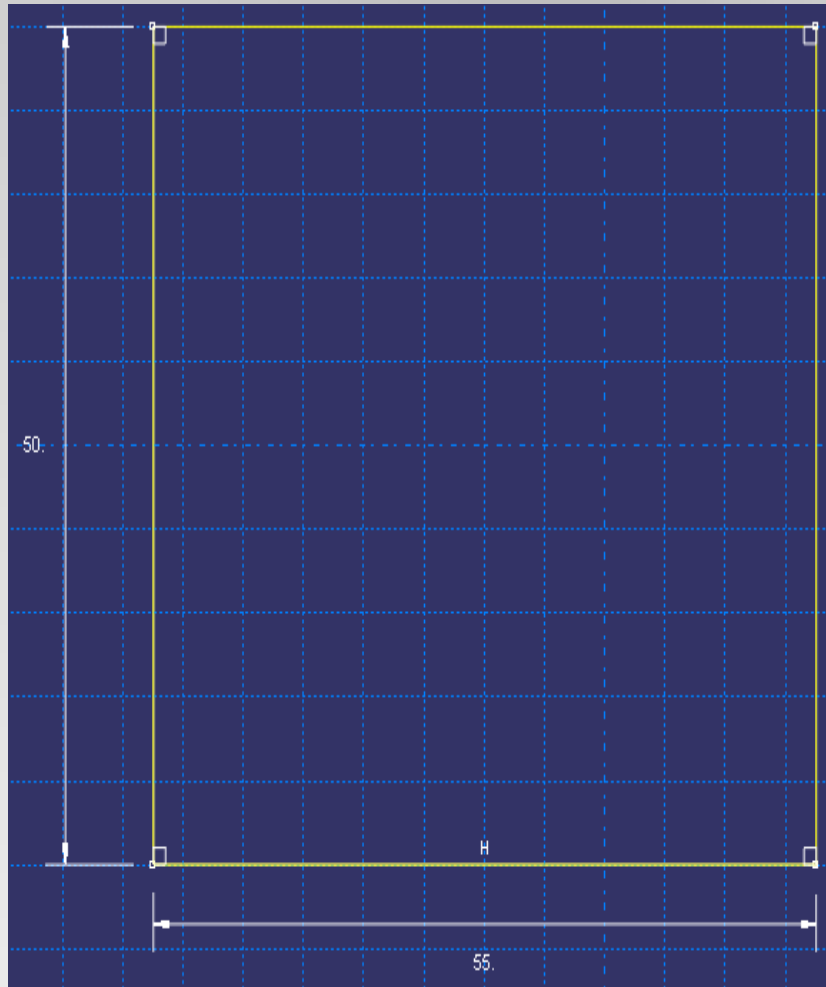
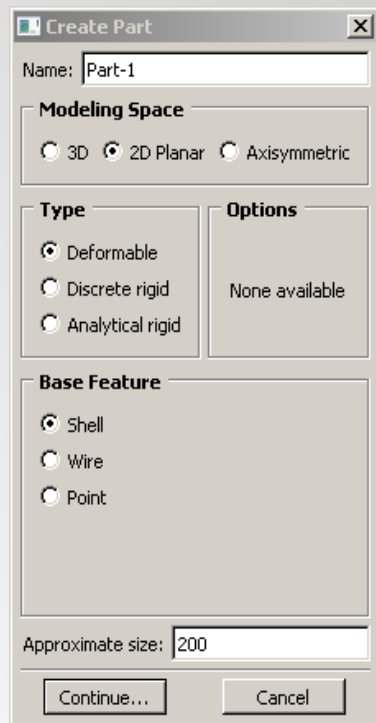


Example 5: Modeling of 2D Particle Reinforced Composite

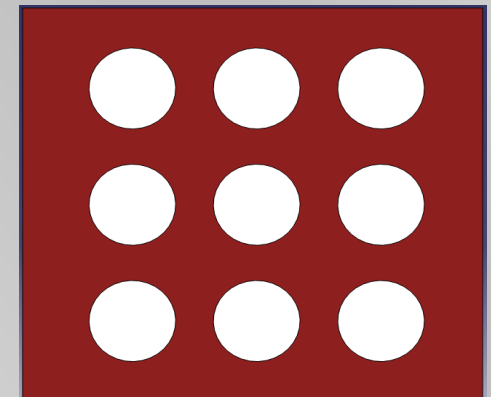
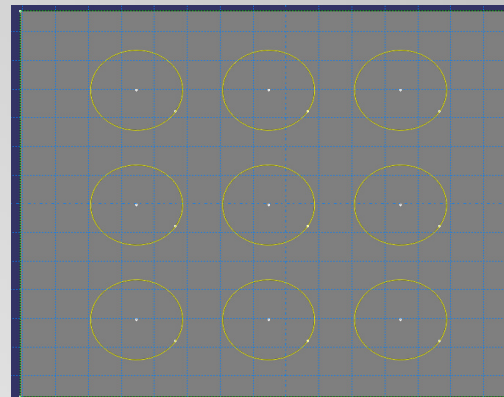
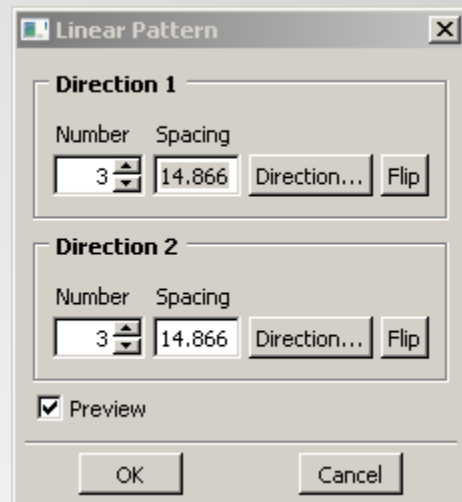
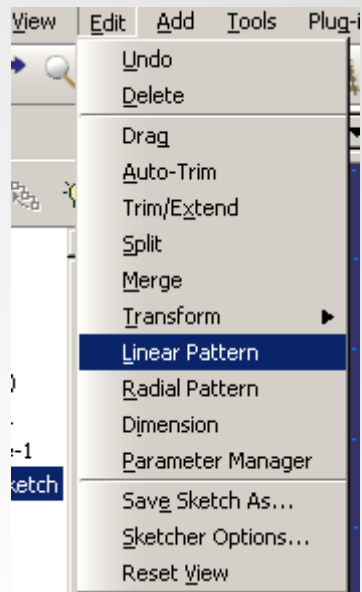
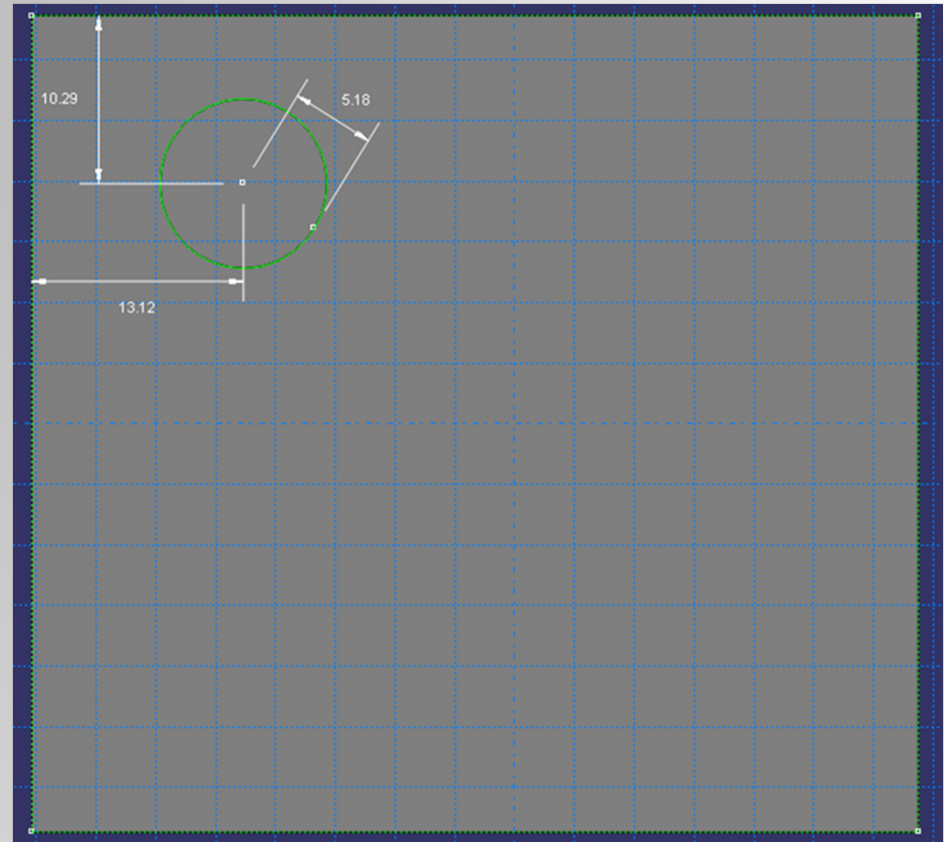
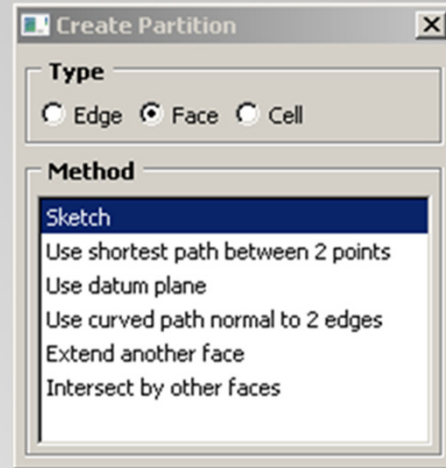
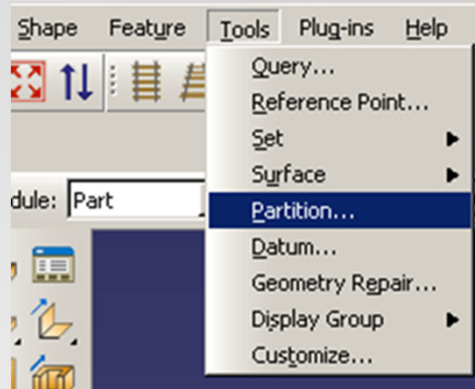
Problem description



Part module



Part module




Property and Mesh module

The image displays several key interfaces from the ANSYS Workbench Property and Mesh modules:

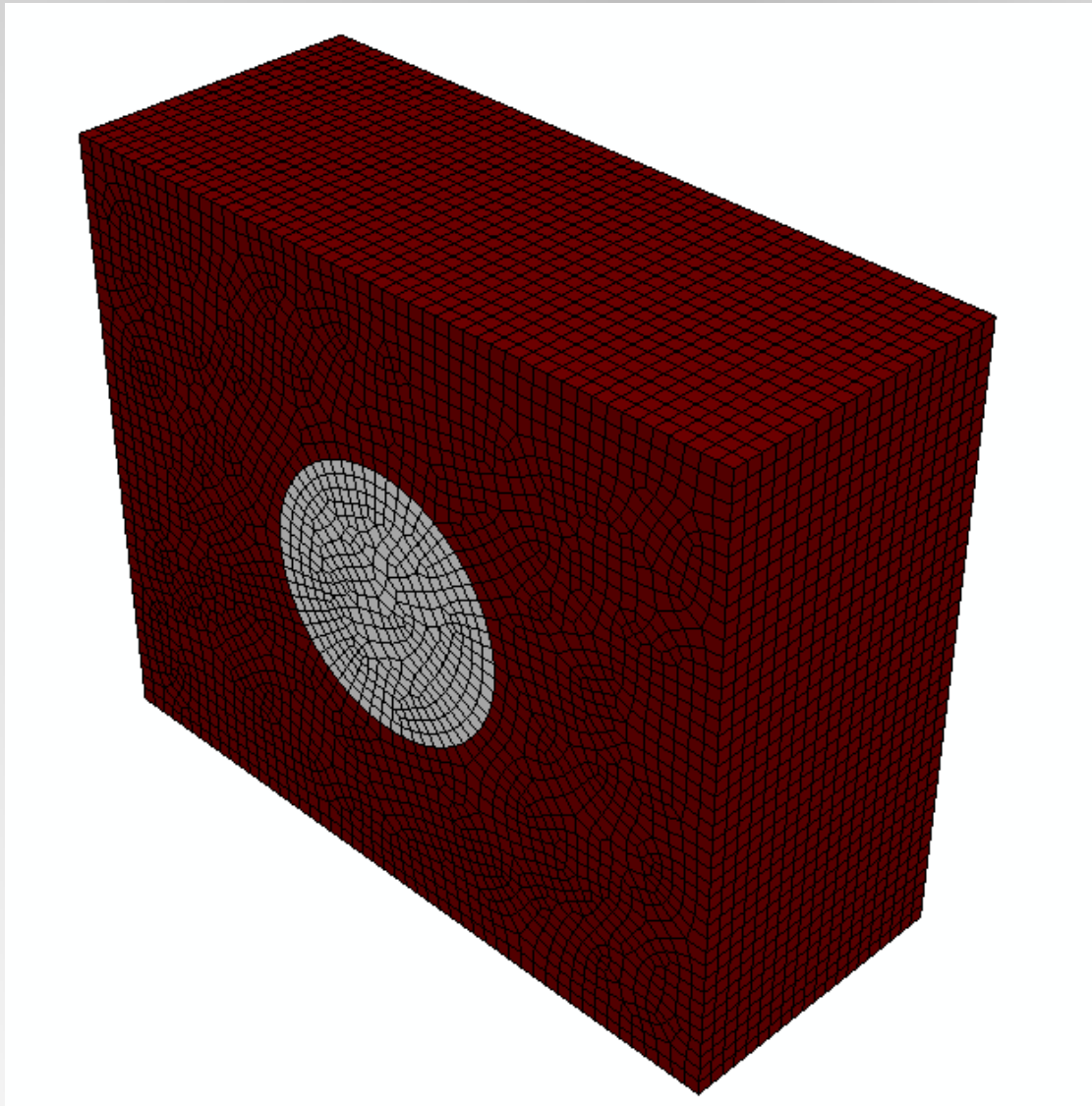
- Create Section Dialog:** Shows 'Section-1' as the name. Under 'Category', 'Solid' is selected. Under 'Type', 'Homogeneous' is selected.
- Edit Section Dialog:** Shows 'Section-1' as the name, 'Solid, Homogeneous' as the type, 'Aluminum' as the material, and '1' as the plane stress/strain thickness.
- Section Assignment Manager:** A table listing assigned sections:

Section Name (Type)	Material Name	Region
Section-1 (Solid, Homogeneous)	Material-1	(Picked)
Section-2 (Solid, Homogeneous)	Material-2	(Picked)
- Mesh Controls Dialog:** Shows 'Element Shape' set to 'Quad'. Under 'Technique', 'Free' is selected. Under 'Algorithm', 'Advancing front' is selected with 'Minimize the mesh transition' checked.
- Mesh Menu:** Shows options like 'Controls...', 'Element Type...', 'Part...', 'Region...', 'Delete Part Mesh...', 'Delete Region Mesh...', 'Create Bottom-Up Mesh...', 'Associate Mesh with Geometry...', 'Edit...', 'Create Mesh Part...', and 'Verify...'.
- Element Size Input:** A field showing 'Element size along the edges (approximate): 0.5'.
- Meshed Model:** A 3D visualization of a meshed part with a grid of circular features.

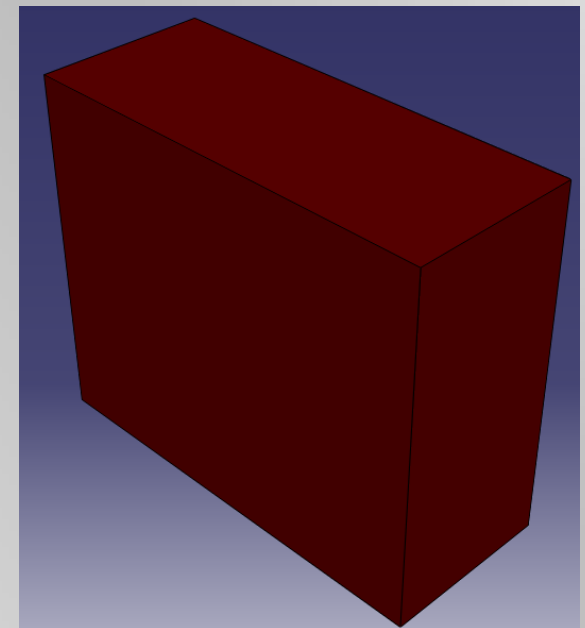
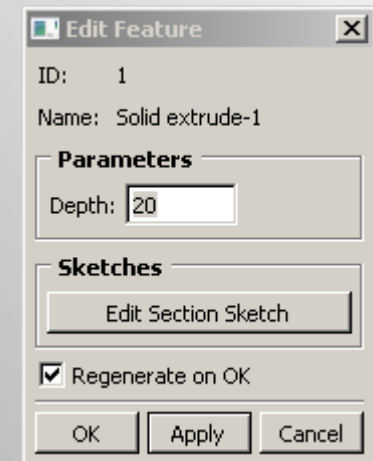
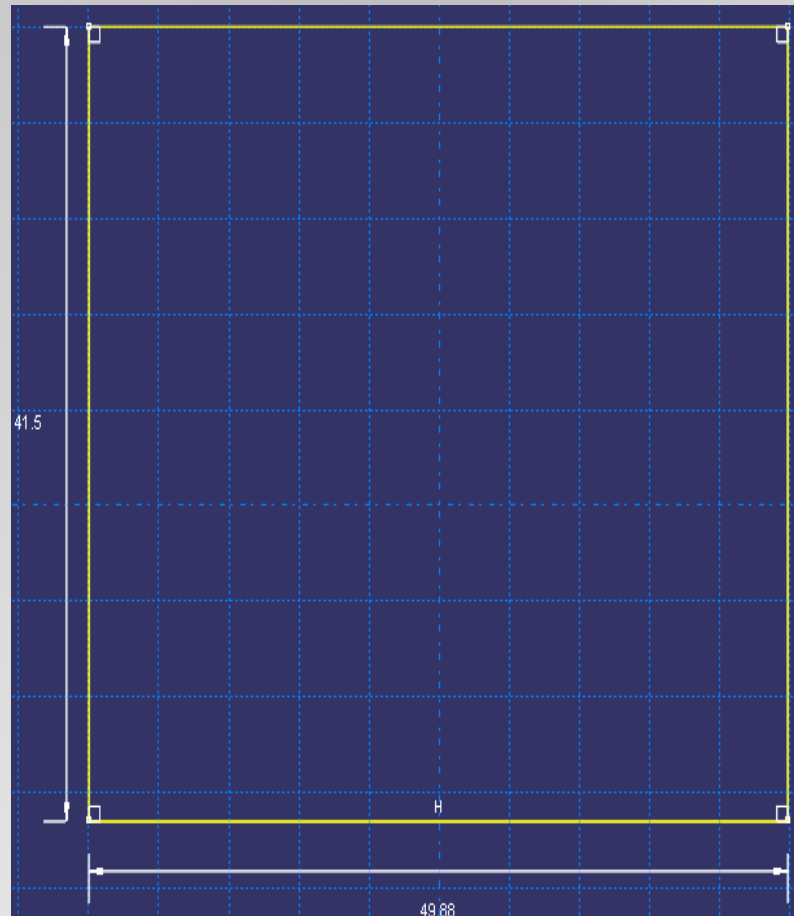
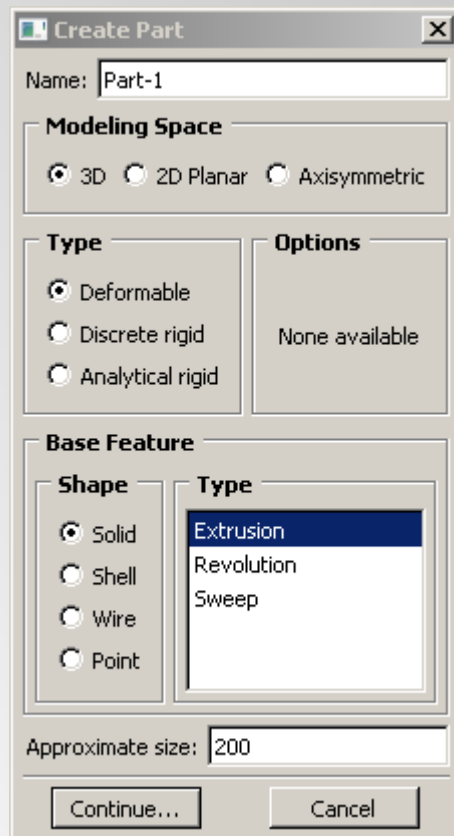


Example 6: Modeling of 3D Fiber Reinforced Composite

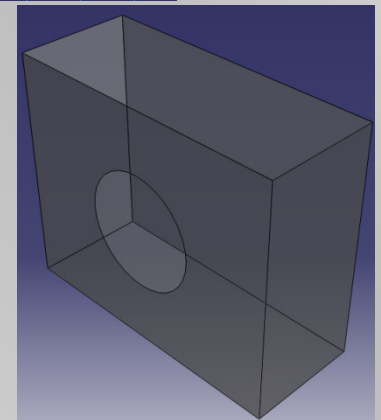
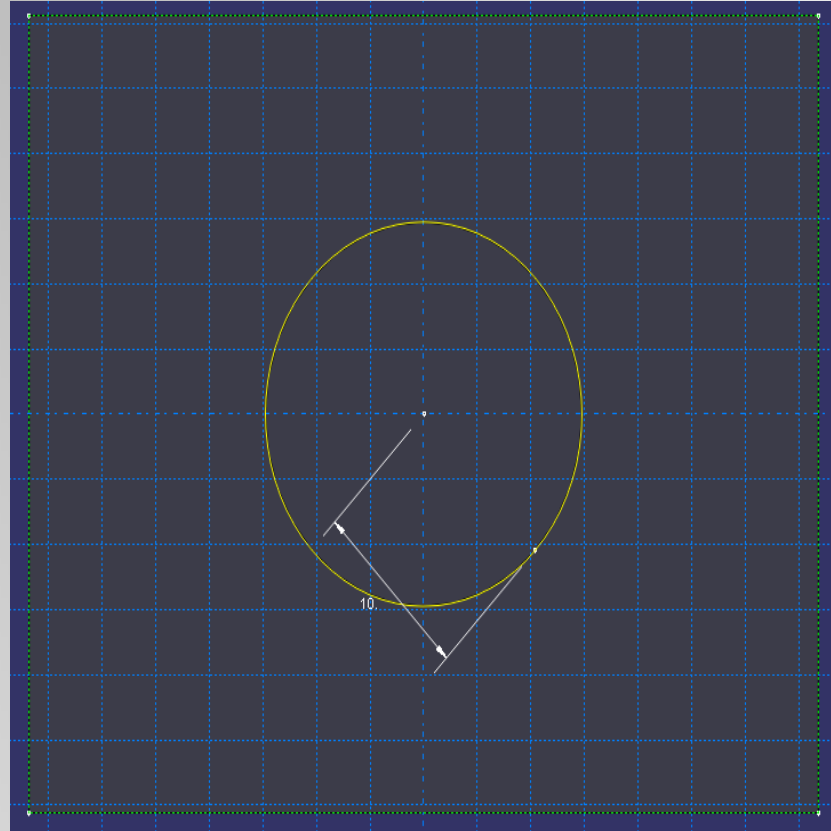
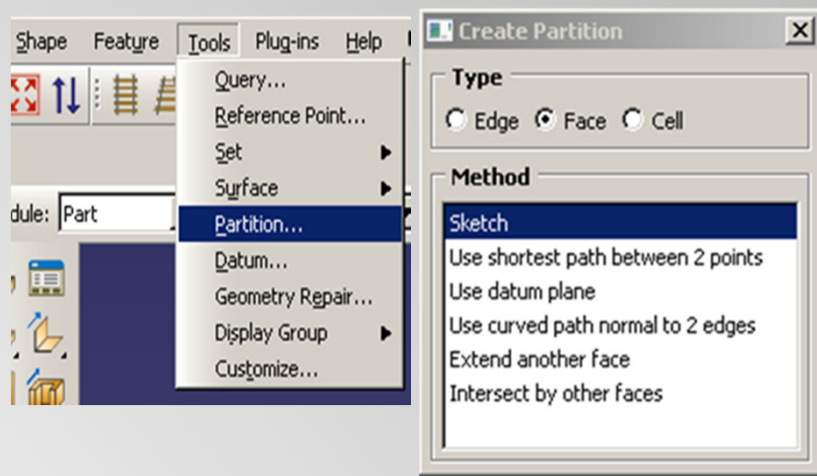
Problem description



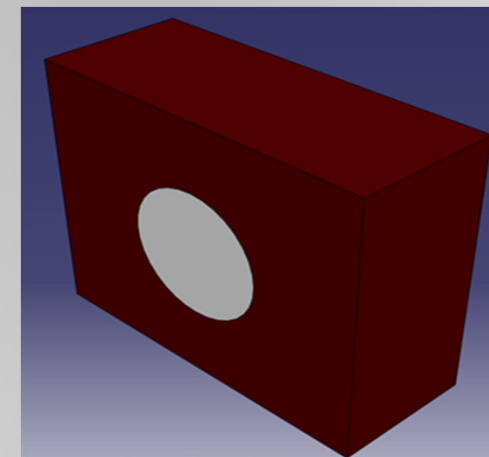
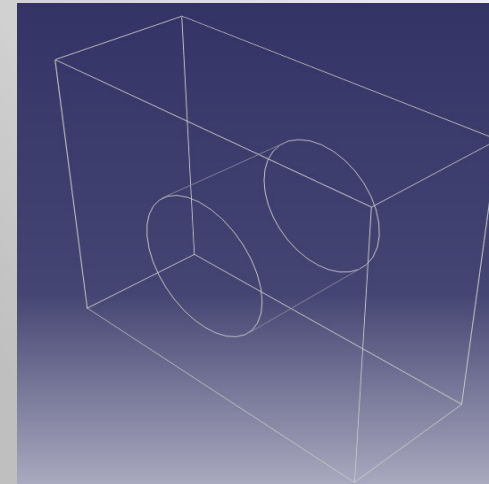
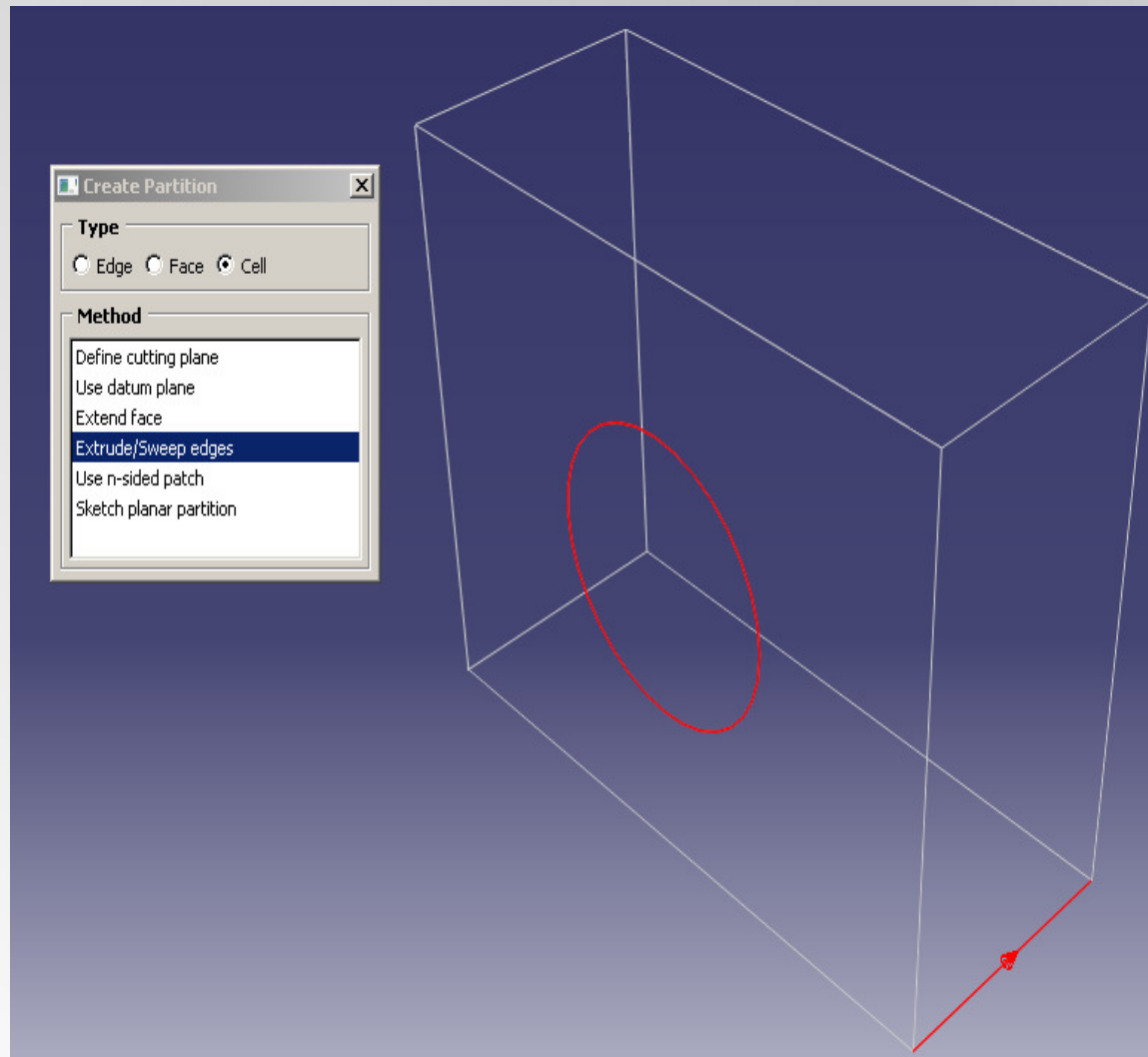
Part module



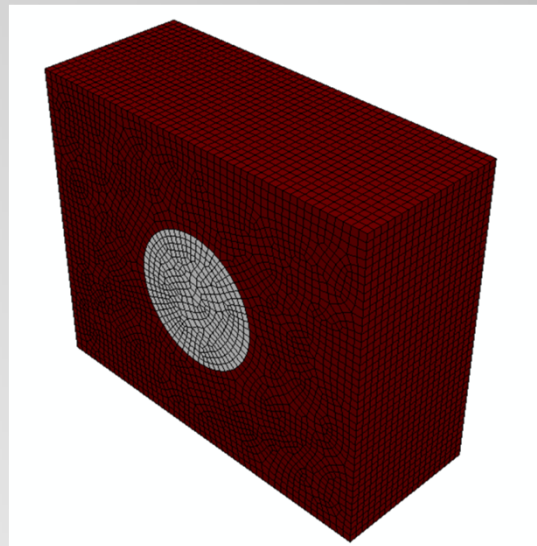
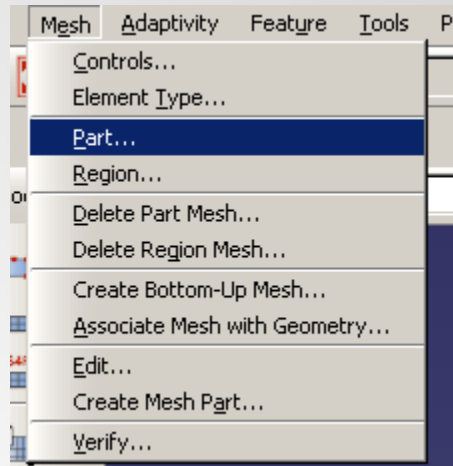
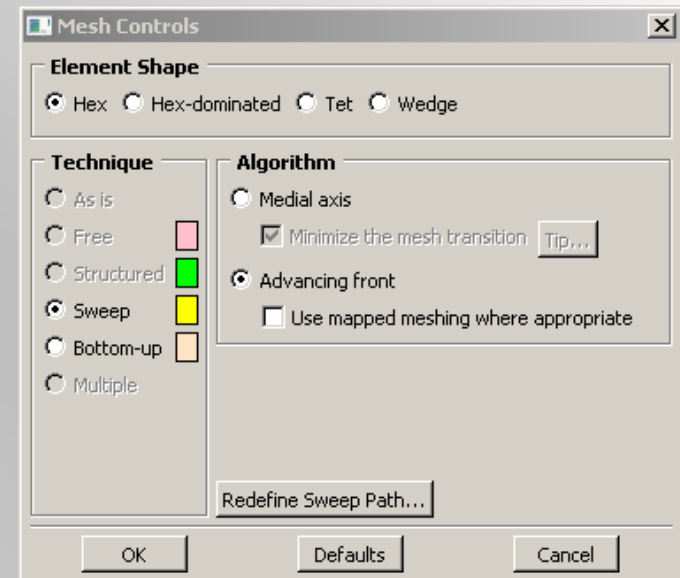
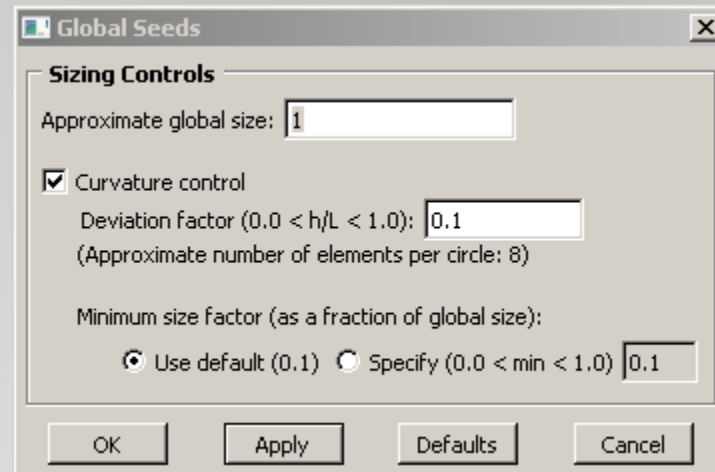
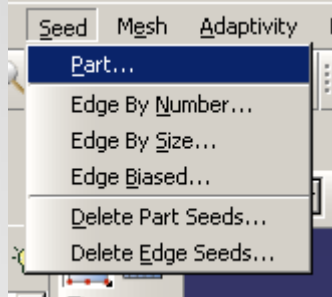
Part module



Part module



Mesh module





References

- 1) ABAQUS manual: <http://sc.tamu.edu:2080/v6.9ef/>
- 2) Matweb, online materials property database: www.matweb.com